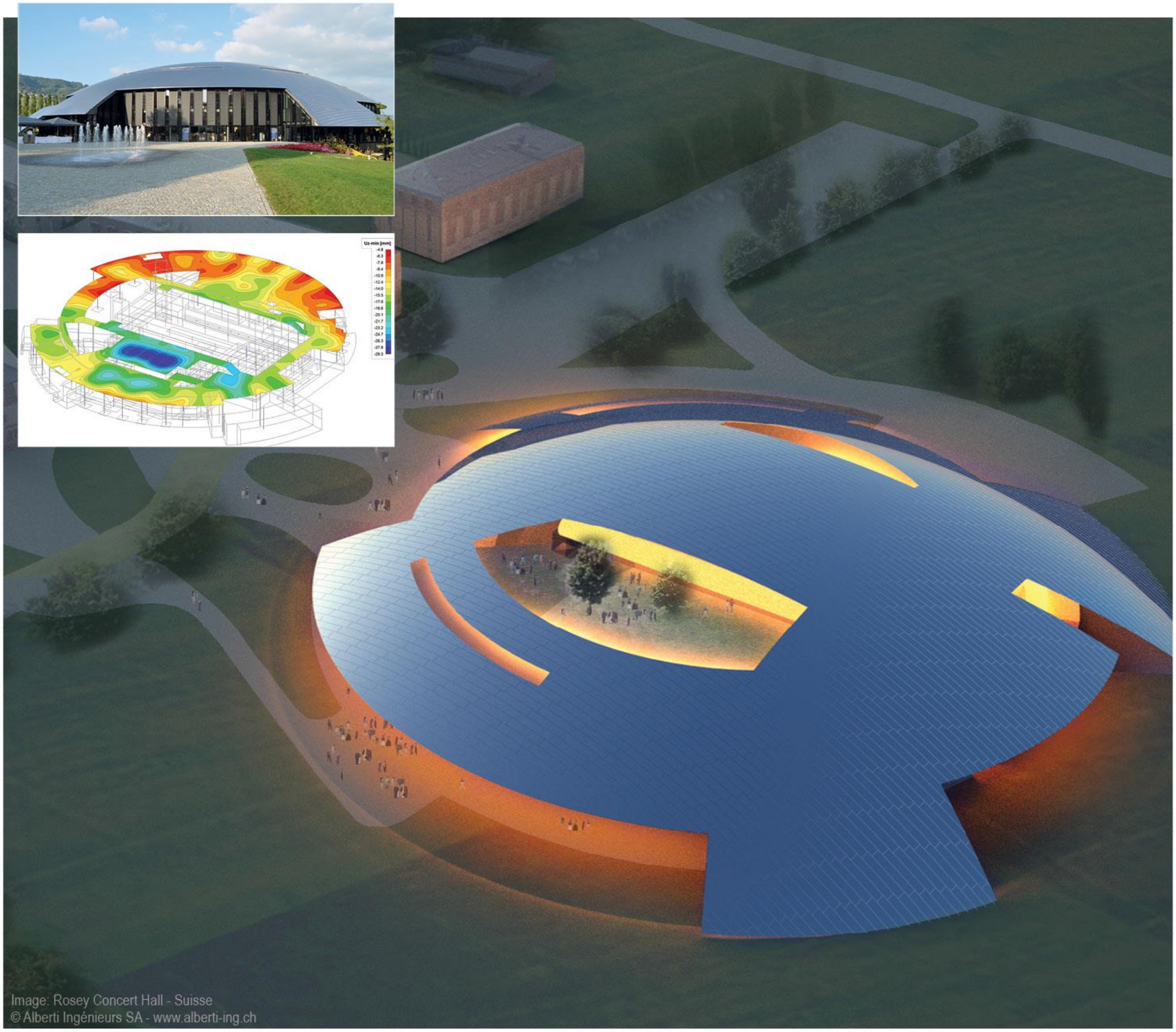
SCIA ENGINEER



Tutorial

Concrete Frame (ACI 318-08)

Table of Contents

General Information

Welcome

Welcome to the Scia Engineer v15 ACI Concrete Frame Tutorial. Scia Engineer is a design program in Windows with a broad application field: from checking/designing simple frames to the advanced design of complex projects in steel, concrete, cold formed steel and a variety of other materials.

The software allows engineers to model 2D and 3D structures which include flat or curved plates and beam members (straight or curved) as well as other advanced 3D geometry. The complete calculation and design process has been integrated into one program so that the input of geometry, input of calculation information (loads, combinations, and supports), linear and non-linear calculation, output of results, reinforcement design according to various codes and the generation of the calculation documentation are all completed in the same software.

Scia Engineer is available in three different editions all which require a license to operate:

- Concept
- Professional
- Expert

License version

A licensed version of Scia Engineer is secured with either a 'dongle', which you apply to the USB port of your computer or a software license on your company's network.

Scia Engineer is also modular and consists of various modules. The user chooses from the available modules and composes a custom design program, perfectly tuned to the desired needs of the company.

In the general product overview of Scia Engineer you will find an overview of the different modules that are available.

Demo version

If the program doesn't find a license on your computer, it will automatically start the demo version. The properties of the demo version are:

- All projects can be inserted however projects created in a demo version cannot be opened in a licensed version.
- The calculation is restricted to projects with 25 elements, 3 plates/shells and two load cases
- The output contains a watermark "Unlicensed software"

Scia Engineer Support

If you need assistance with the software, you can contact the Scia Engineer support service in the following manners:

By e-mail

Send an e-mail to support@scia.net with a description of the problem and the concerning *.esa file, and mention the number of the version you are currently working with.

By telephone

From USA: 443-542-0637

Via the Scia Support website

http://www.scia-online.com/en/online-support.html

Website

Link to Tutorials

http://www.Scia-online.com > Support & Downloads > Downloads > input e-mail address > Scia Engineer > Scia Engineer Manuals & Tutorials

Link to eLearning

http://www.scia-online.com > Support & Downloads > eLearning

Link to Demo version

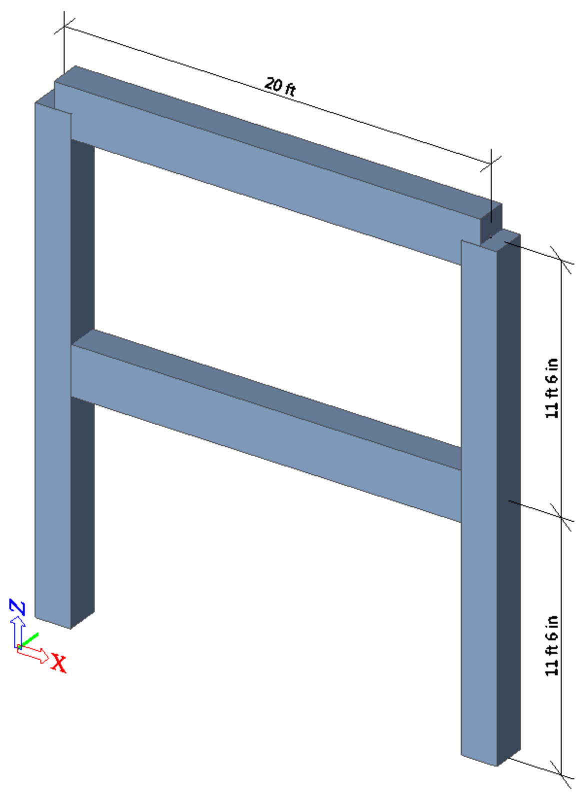
http://www.Scia-online.com > Support & Downloads > Secured Downloads > input username and password > Service Packs > Scia Engineer > Setup – Scia Engineer

Introduction

This tutorial describes the main functions of Scia Engineer 15, the input of design data and calculation of a 2D concrete frame.

The tutorial will begin with the creation of a new project and the modelling of the concrete frame structure. After the input of all frame geometry and loads, the structure will be calculated and the results will be viewed. Next, the input of the slenderness and buckling parameters of the frame will be discussed, followed by the concrete calculations which contain the inclusion of longitudinal, transverse and additional reinforcement. Lastly, the tutorial will discuss how to format a calculation document while properly displaying the calculation results.

The figure below shows the computational model of the structure that will be completed through this tutorial (the units are in the Imperial system, e.g. kips, inch):



Getting started

Starting a project

Starting the program

Before you can start a project, you need to start the program first.

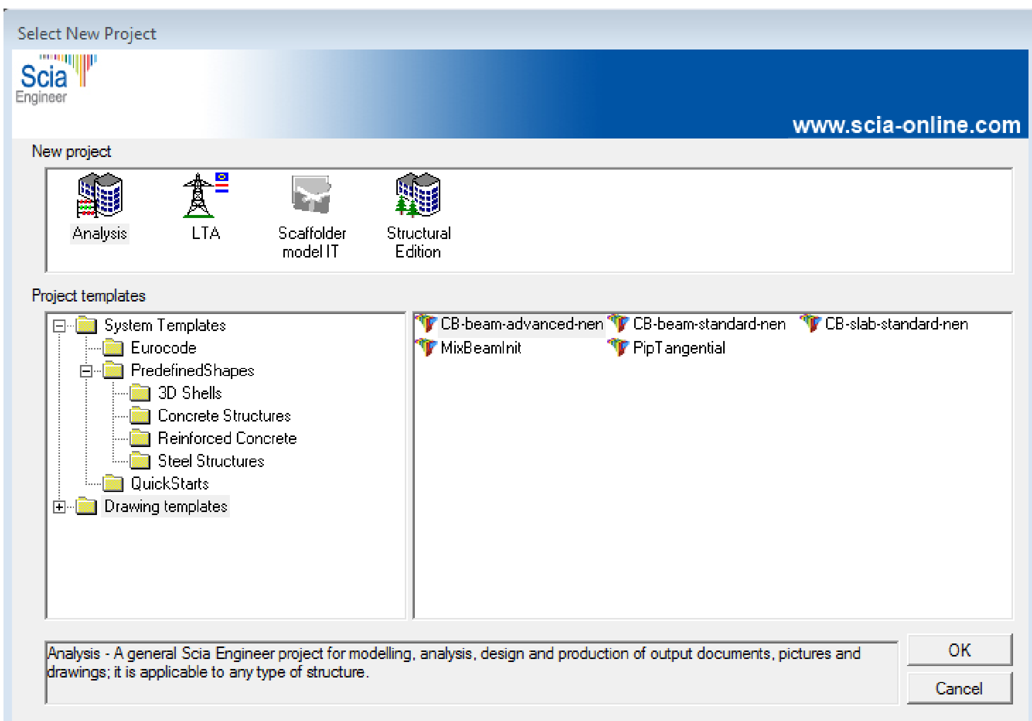
1. Double-click on the Scia Engineer shortcut in the Windows Desktop.
   Or:
2. If the shortcut is not installed, click **[Start]** and choose **Programs > Scia Engineer 2013.1 > Scia Engineer 2013.1**.

If the software cannot locate a license file, you will receive a dialogue box indicating that no license was found. A second dialogue box will then list the restrictions of the demo version. Click **[OK]** in both windows.

For this Tutorial, you must start a new project.

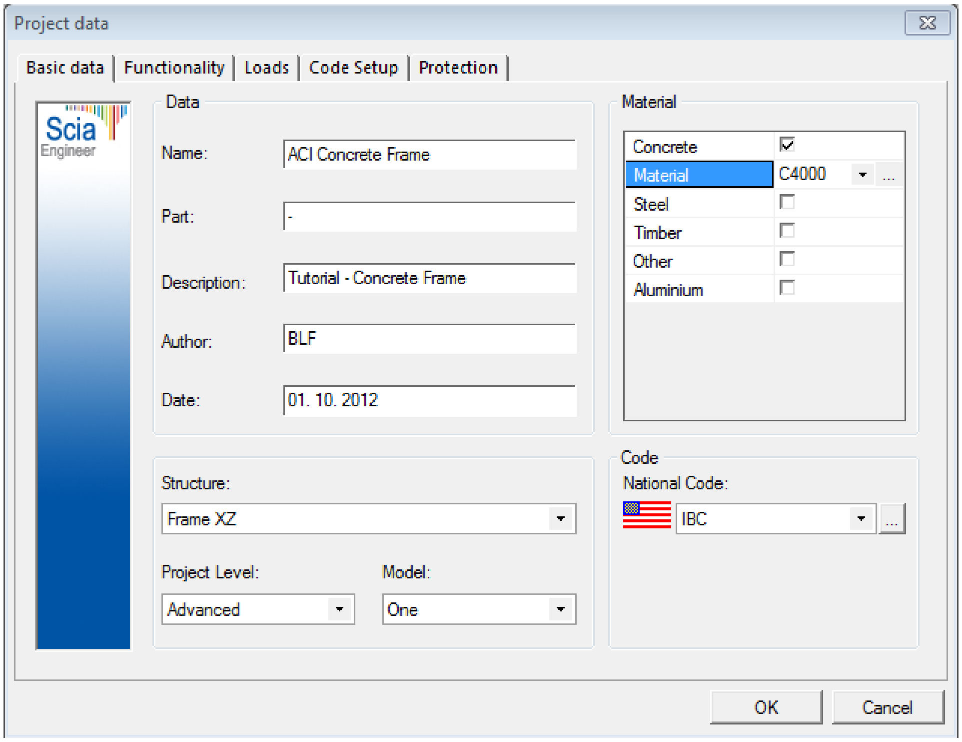
Starting a new project

1. If the program shows the **Open** dialogue box, click **[Cancel]**.
2. Click the **New** icon in the toolbar.



In the **Select New Project** dialogue box, choose the **Analysis** environment by clicking on the corresponding icon. Confirm your choice by clicking **[OK].**

Now, the **Project data** dialogue box is opened. Here, you can enter general data about the project.



3. In the **Basic Data** group, enter your preferred data. The data you enter will be displayed on the output produced by Scia Engineer, e.g. in the document and on the drawings.
4. Choose the **Project level: Advanced** and **Model: One**.
5. Click on the rectangular button ... below **National Code** to choose the default code for the project. This code will determine the available materials, combination rules and code checks. For the tutorial we will choose IBC (International Building Code). The window **Codes in project** is opened.

   - Click **[Add]**.
   - The dialogue box **Available national codes** are opened.
   - Select the USA flag and click **[OK]**.
   - You will return to the **Codes in project** dialogue and **IBC** is added.
   - Select the flag named **IBC.**
   - Select the **Active code** option and click **[Close]**.
   - You will return to the **Project data** window and **IBC** is the active code.

6. Select **Frame XZ** in the **Structure** field.

   The structure type (Frame XZ, Frame XYZ, Plate XY, General XYZ, etc.) will restrict the input possibilities during the calculation.

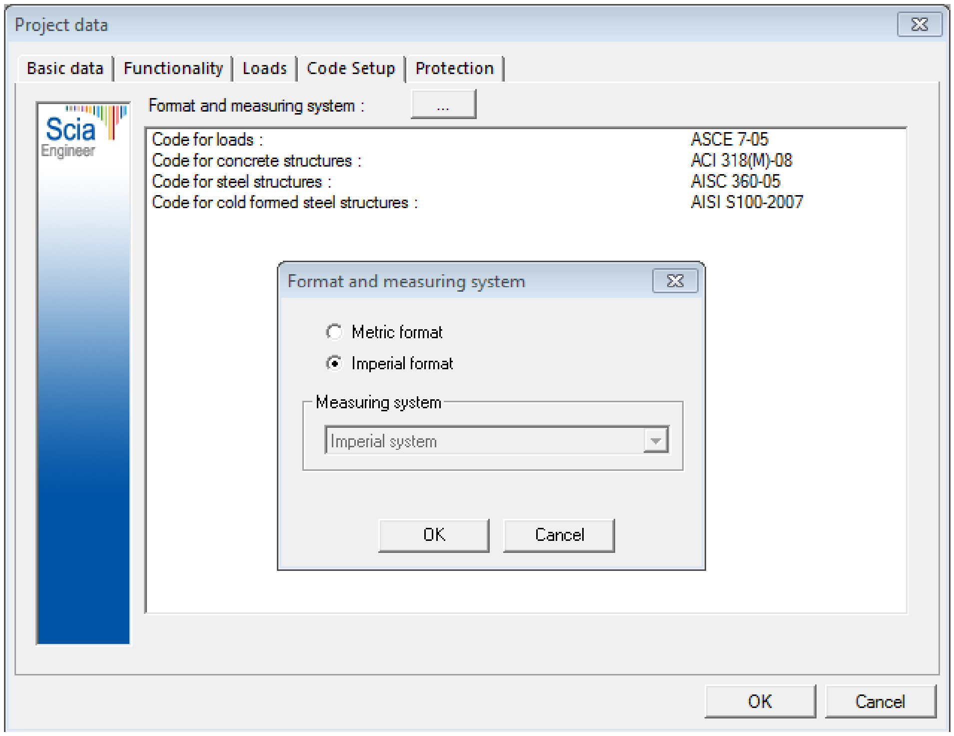
7. In the **Material** group, select **Concrete**. Below the item **Concrete**, a new item **Material** will appear.

8. Choose **C4000** from the menu. This choice corresponds with the compressive design strength of the concrete.

9. Confirm your input with **[OK]**.

10. On the **Code Setup** tab, you can see the specific codes that are used for the generation of loads and the design of steel, concrete and cold formed steel structures. In addition to these codes, you can also choose the measurement system and formatting that will be used throughout the project.

11. Click on **Code Setup** tab and click on the ... button beside **Code setup**. Now, the **Format and measuring system** dialogue box is opened. Choose **Imperial format** and click **[OK].**



<u>Notes:</u>

*On the **Basic data** tab, you can set a project level. If you choose "standard", the program will only show the most frequently used basic functions. If you choose "advanced", all basic functions will be shown.*
*On the **Functionality** tab, you choose the options you need. The non-selected functionalities will be filtered from the menus, thus simplifying the program and the analysis.*
*On the **Load** tab, you will find the value for the acceleration of gravity and the applicable wind and snow loads that can be activated with the Climate loads functionality.*

Project management

Save, Save as, Close and open

Before entering the software to complete the frame construction, we will first discuss how to save a project, how to open an existing project and how to close a project. While completing the tutorial, the project can be saved at any time, that way you can leave the program at any time and resume the project from the save location later.

Saving a project

Click on  in the toolbar.

If a project has not yet been saved, the dialog box **Save as** appears. Navigate to the location or the drive where you want to save your project in. Select the folder or subfolder in which you want to save the project and enter the project file name in **File name** field. Once this is complete click on **[Save]** to save the project.

If you press  twice, the project is automatically stored with the same name. If you choose **File > Save as** in the main menu, you can enter a new file name or save location for the project file.

Closing a project

To close a project, choose **File > Close** in the main menu.

A dialog box appears asking if you want to save the project. Depending on your choice, the project is saved and the active dialog box is closed.

Opening a project

Click on  to open an existing project.

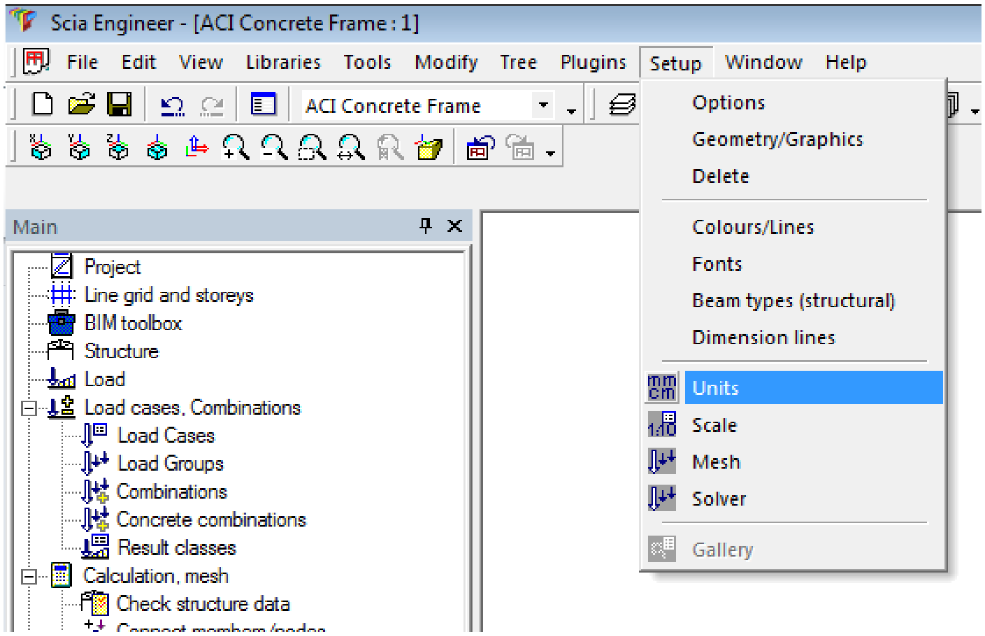
A list with Scia Engineer projects appears. Select the desired project and click **[OK]** (or double-click on the project to open it).

Units

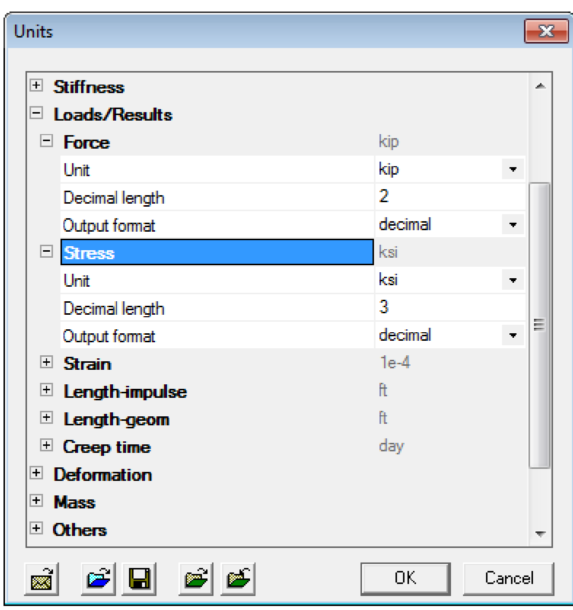
Changing the units

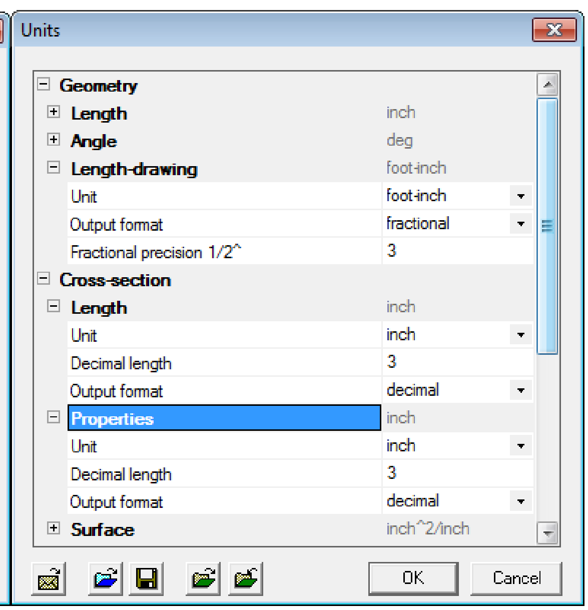
At the beginning of a new project, verify that the units used by Scia Engineer are compatible with your desired units. To check and modify the units:

1. Click on **Setup** in the main menu and select **Units**.



2. In the Units window you can view the units for different options and modify them to your desired unit system.





Note:

It is possible to create a default template file that contains user preferences for desired units, scales and fonts. A template file can be created and saved so that the user can load the desired template file before beginning a new project. The creation and use of template files is not discussed in this tutorial.

Geometry input

Input of the geometry

When starting a new project, the specific geometry of the structure must be entered. The structure can be entered directly, or it is possible to add geometry using instance templates with parametric blocks, DXF files, DWG files and other formats.

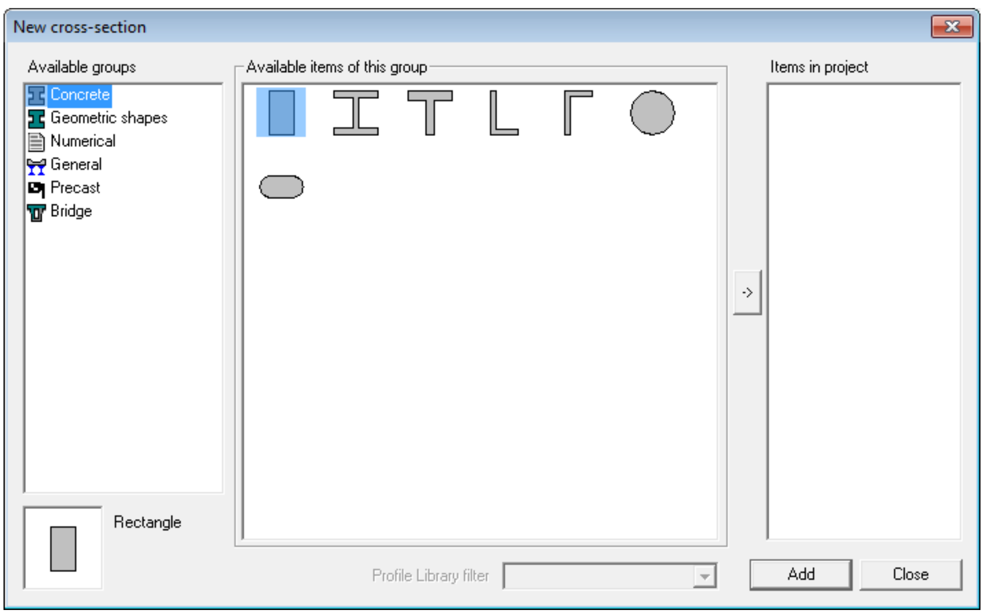
Profiles

When entering one or more 1D structural elements, a profile type is immediately assigned to each member. By default, the active profile type is represented. At any time it is possible to open the profile library to activate another profile type. If you want to add a structural part before a profile type has been defined, the profile library will automatically be opened.

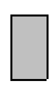
Adding a profile

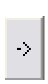
1. Click on the **Cross-Sections**  icon in the toolbar.

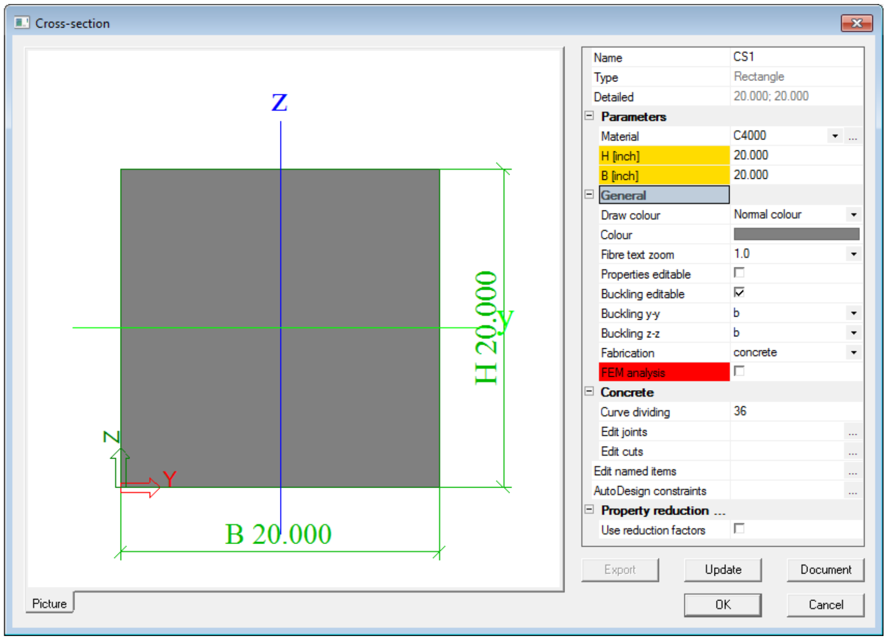
   The **Cross-Section manager** is opened. If no profiles have been entered in the project, the **New cross-section** window will be automatically opened.



2. Click **Concrete** in **Available groups**.

3. In the **Available items of this group**, you can choose a rectangular section  .

4. Click **[Add]** or  to add the profile to the project. After adding, the **Cross-Section** window appears.



5. In this window, you can change the properties of the rectangular section. Enter **20 inch** for height **H** and **20 inch** for width **B**.

6. Click **[OK]** to confirm, the profile is added to the **Items in Project** group.

7. A second rectangular section with height **H = 28 inch** and width **B = 18 inch** is added in a similar way.

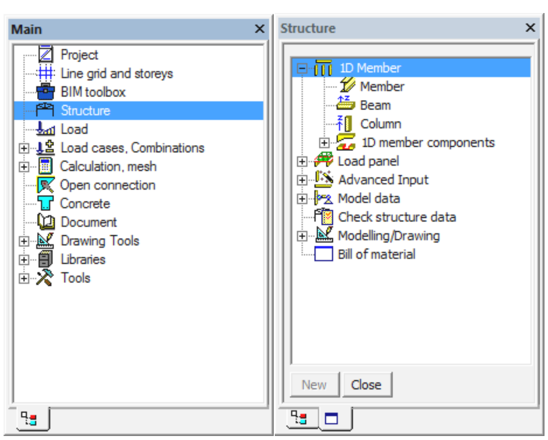
8. Click **[Close]** in the **New Cross-Section** window, the **Cross-Sections** Manager appears.

9. Click **[Close]** to close the **Cross-Section** Manager and to return to the project.

Geometry

Structure menu

1. When a new project is started, the **Structure menu** is automatically opened in the **Main window**. If you want to modify the structure at a later time, you must double-click on **Structure** in the **Main window** to activate the menu.

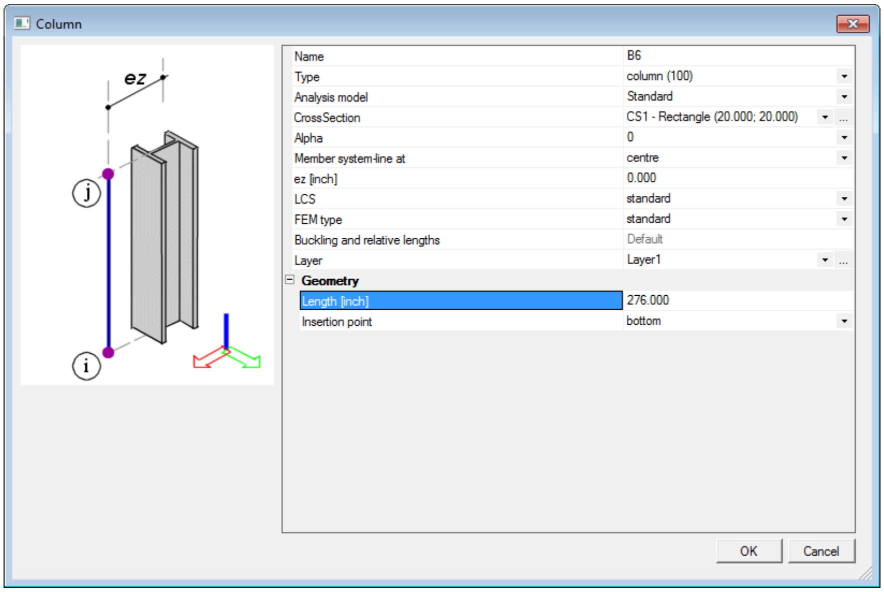


2. In the **Structure menu**, you can choose different structural elements to enter into the structure.

To model the frame, you first must enter the columns. After the columns have been entered, the beams on top and in the middle of the columns can be entered.

Entering a column

1. Use the **Column** option in the **Structure Menu** to enter a new column.



2. In the **Cross-section** field, choose the concrete section **CS1 – Rectangle (20.000; 20.000)**

3. Specify the column length as **276 inch or (23'-0")**.

4. The insertion position is by default set to **Bottom** so that the bottommost point determines the position of the column.

5. Confirm your input with the **[OK]** button.

6. The first column in the frame is positioned at the origin of the coordinate system. To accomplish this, you must enter the coordinates **0, 0** in the **Command line** and then press **<Enter>** to confirm your input.

7. A second column is entered in a similar way at position **240,0 or (20', 0')**

8. End the input using the **<Esc>** key.

9. After input of an entity in Scia Engineer, the entity is always selected. In this instance, the columns are colored magenta meaning they are the active selection. To cancel the selection, press the **<Esc>** key once more.

Notes:

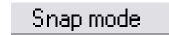
*The properties of selected elements are shown and can be modified in the **Properties window**. If no section has been defined in the project, the **New cross-section** window will automatically appear as soon as you try to enter a structural element (column, beam, brace, etc.). At any time, you can end your active input by pressing either the **<Esc>** key either the right mouse button.*

*To quickly visualize the entire structure, click **Zoom All***  *in the toolbar.*

When entering coordinates using the command line, separate the desired coordinates using either a comma (,) or a space.

When both columns are entered into the model, you can start adding the beams to the frame. The start and end points of the beams are already known, i.e. the center and the end point of the column. Therefore, the beams will not be entered through coordinates, but using the **Cursor Snap Settings.**

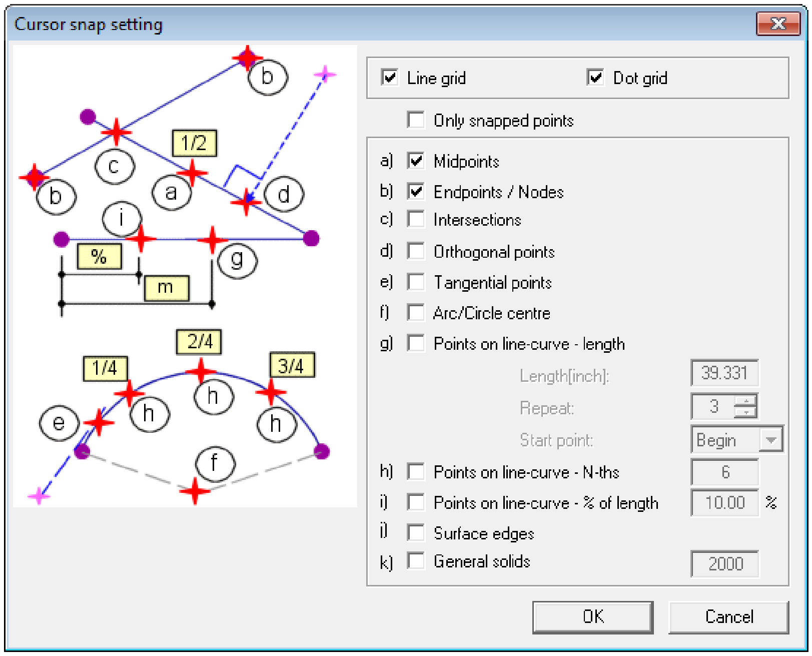
Cursor snap settings

1. Click on the **Cursor snap settings** icon  in the Command line or click on the button 

at the lower right of the screen. The **Cursor snap settings** window is opened.

2. Activate the options a) and b) to select the midpoints and the end points of members as snap points in this project.

3. Click **[OK]** to confirm.

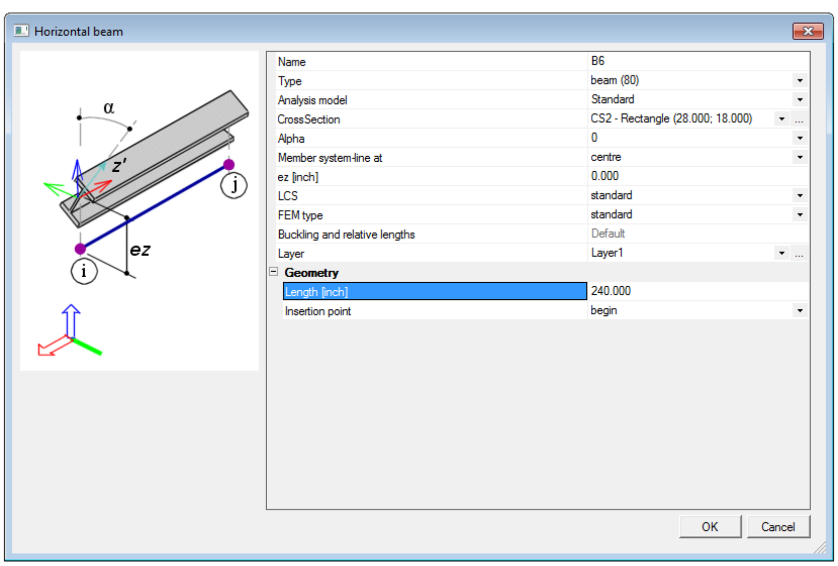


Now, you can enter the concrete beams.

Entering a beam

1. To enter a new beam, you use the **Beam** option in the **Structure menu.**

2. In the **Cross-section** field, choose the second section **CS2 – Rectangle (28.000, 18.000)**

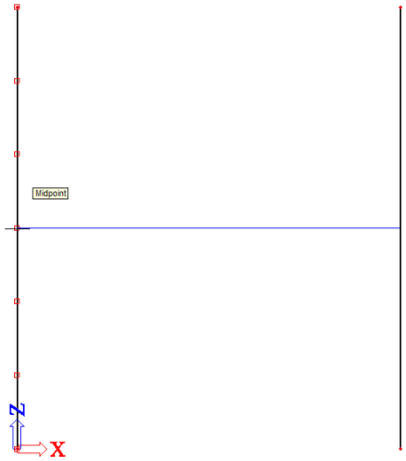


3. Enter the beam length as **240 inch or (20'-0")**.

4. The insertion position is by default set to **Begin** so that the left point determines the position of the beam.

5. Confirm your input with **[OK]**.

6. Now click with the mouse on the center of the left-hand side column to enter the beam:



7. The upper beam is entered in a similar way by clicking the top node of the left-hand side column.

8. Press **<Esc>** to terminate the input.

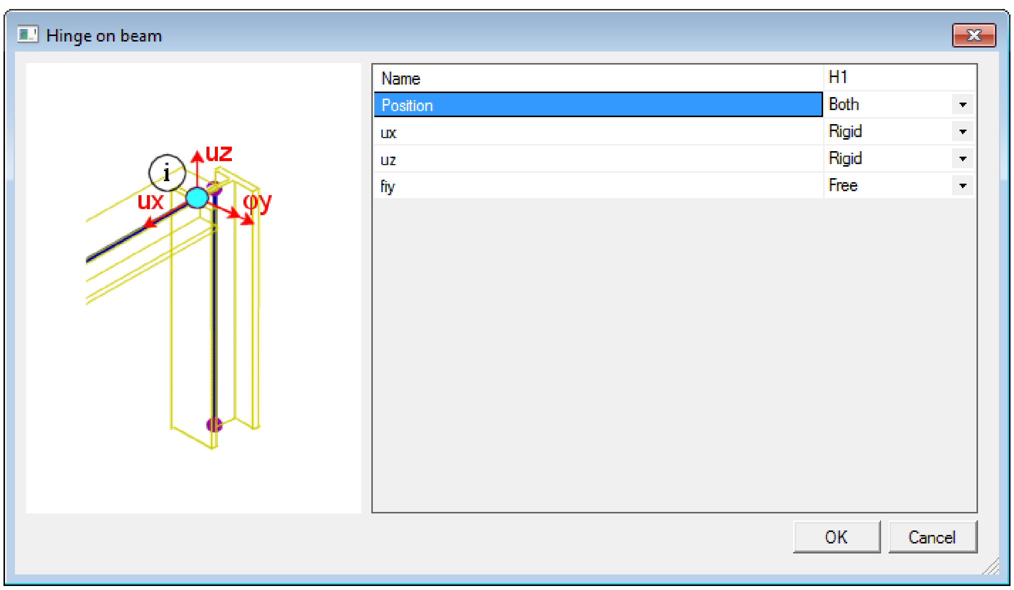
9. Press **<Esc>** once more to cancel the selection.

Hinges

In this project, the beams are connected to the columns in a hinged manner. Since the structure type has been chosen as Frame XZ, by default the members will be connected to each other with fixed ends. Because of this, hinges must be manually added to the frame.

Entering hinges

1. To enter hinges, use the **Model data > Hinge on beam** option in the **Structure menu**.



2. The hinges are needed at both the beginning and the end of the beam, therefore the **Both** is selected as the **Position** of the hinge.

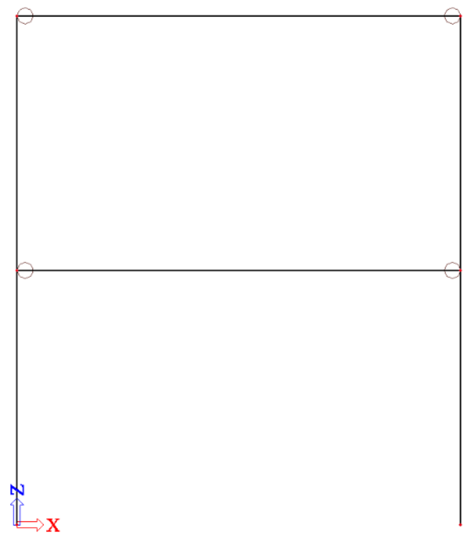
3. To obtain a hinge, the rotation **fiy** is taken **Free**, while the translations remain **Fixed**.

4. Confirm your input with **[OK]**.

5. The hinges are added by clicking with the left mouse button on both the top and bottom beams.

6. Press **<Esc>** to terminate the input.

7. Press **<Esc>** once more to cancel the selection.

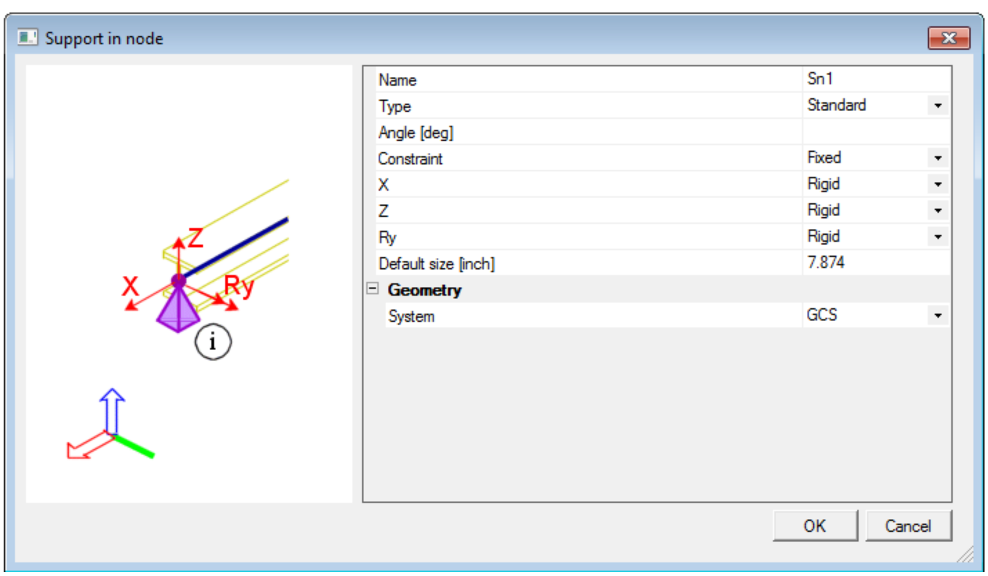


Supports

The geometry input can be completed with supports. Both column bases are modelled as fixed supports.

Entering supports

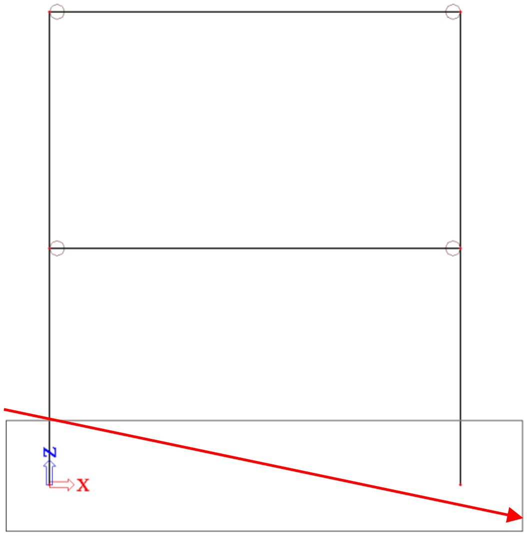
1. To enter supports, use the **Model data > Support > in node** option in the **Structure menu.**



2. To model the ends as fixed, the **Constraint** is set to **Rigid**. This will set the rotation of the support in the applicable directions based on the structure type of the project (Frame XZ) to Rigid.

3. Confirm your input with **[OK]**.

4. To specify support type on the frame, simply select both bottom nodes by drawing a box with the mouse from the left-hand side to the right-hand side:



5. Press **<Esc>** to terminate the input.

6. Press **<Esc>** once more to cancel the selection.

<u>Notes:</u>

If you draw the box from the left-hand side to the right-hand side, only entities, which are completely in the rectangle, will be selected. If you draw the rectangle from the right-hand side to the left-hand side, the entities, which are completely in the rectangle, as well as the entities that intersect with the rectangle will be selected.

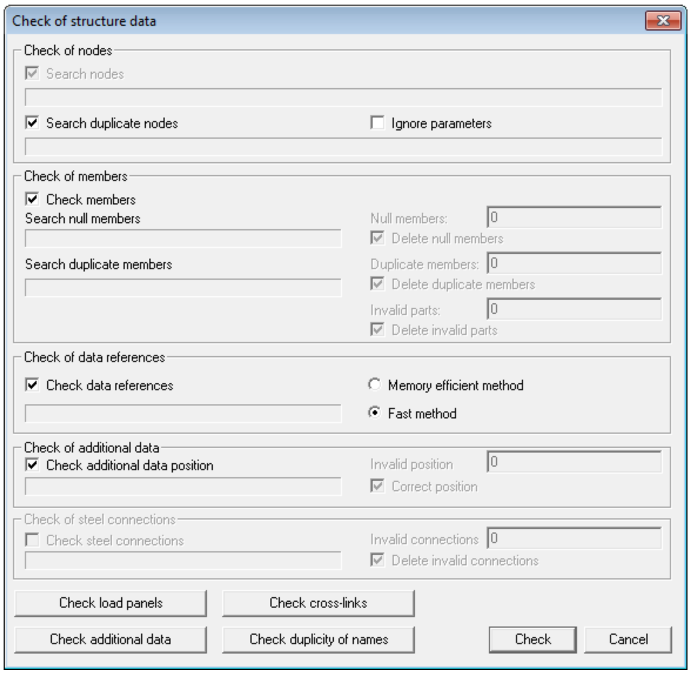
*The **Command line** includes a number of predefined supports. For this project, you could also have used the **Fixed Support***  *icon.*

Check Structure data

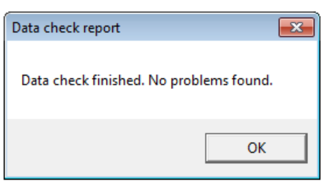
After input of the geometry, the input can be checked for errors by means of the option **Check Structure data**. With this tool, the geometry is checked for duplicate nodes, zero beams, duplicate beams, duplicate hinges and duplicate supports.

Checking the structure

1. Double-click on the **Check Structure data** option in the **Structure Menu** or click on the  icon in the toolbar.
2. The **Structure data check** window appears, listing the different available checks.



3. Click **[Check]** to perform the checks.
4. The **Data Check Report** window appears, indicating that no problems were found.



5. Close the check by clicking **[OK]**.

Connecting entities

The start and end node of the top beam is an end node of a column. Therefore, this beam is automatically connected to the columns.

The beam in the middle of the columns, however is not connected at a column end node. The end nodes of the beam are located at some internal point within the column and therefore are not yet connected to the columns. In this paragraph, we will explain how to connect the members to each other.

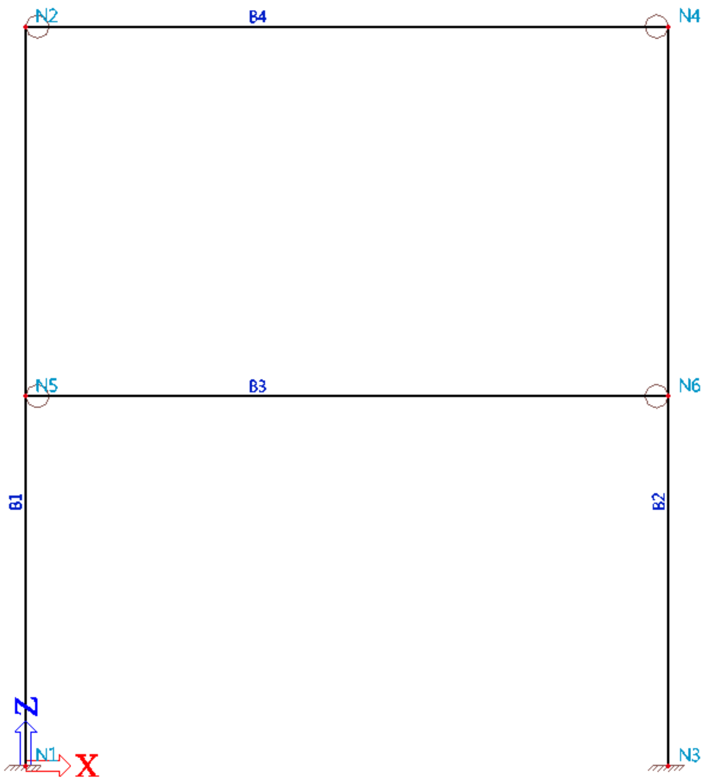
To display the names of the bars and nodes, you can activate the labels by means of the buttons in the **Command line**.

Activating node labels

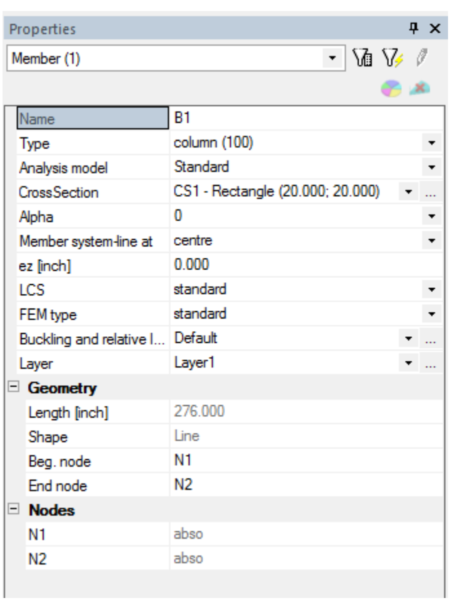
Node labels are activated by means of the  icon on top of the **Command line**.

Activating member labels

Member labels are activated by means of the  icon on top of the **Command line**.



When you select column **B1** with the left mouse button, the properties are displayed in the **Property Window**:



Properties

Member (1)

Name	B1
Type	column (100)
Analysis model	Standard
CrossSection	CS1 - Rectangle (20.000; 20.000)
Alpha	0
Member system-line at	centre
ez [inch]	0.000
LCS	standard
FEM type	standard
Buckling and relative l...	Default
Layer	Layer1
Geometry	
Length [inch]	276.000
Shape	Line
Beg. node	N1
End node	N2
Nodes	
N1	abso
N2	abso

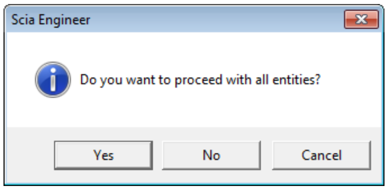
This window indicates that the begin node is **N1** and the end node **N2**. Node **N5** is not part of the column. To connect beam **B3** to the columns, you must use the option **Connect members/nodes**.

Connecting entities

1. Press **<ESC>** or click the **Cancel selection**  icon to deactivate any selection of entities.

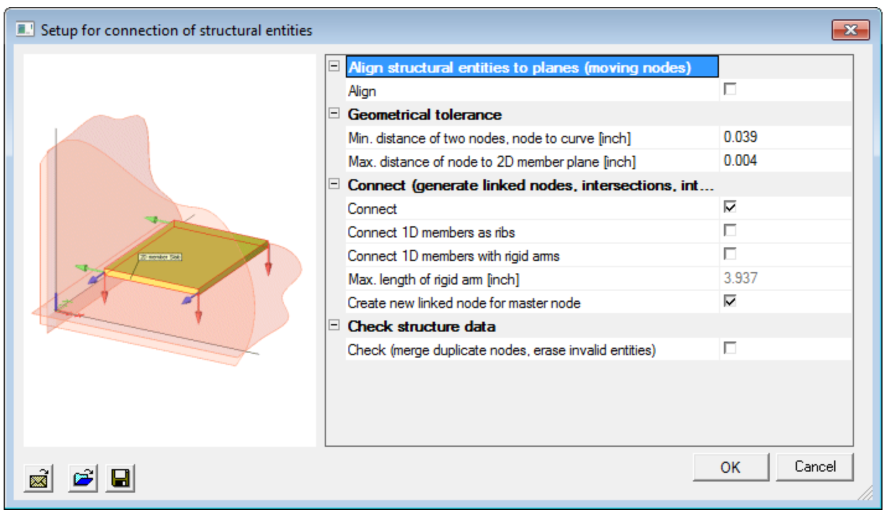
2. Double-click on the **Model data > Connect members/nodes** option in the **Structure menu** or click the  icon in the toolbar.

3. A dialogue box appears asking if all nodes should be connected.



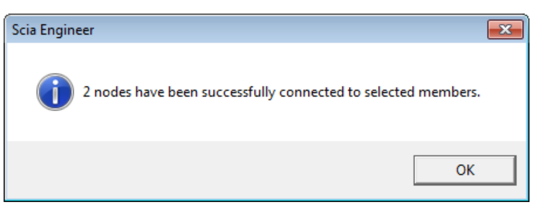
4. Click **[Yes]**.

5. The **Setup for connection of structural entities** dialogue box now appears.

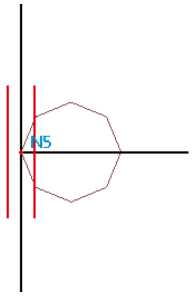


6. Confirm the settings by clicking **[OK]**.

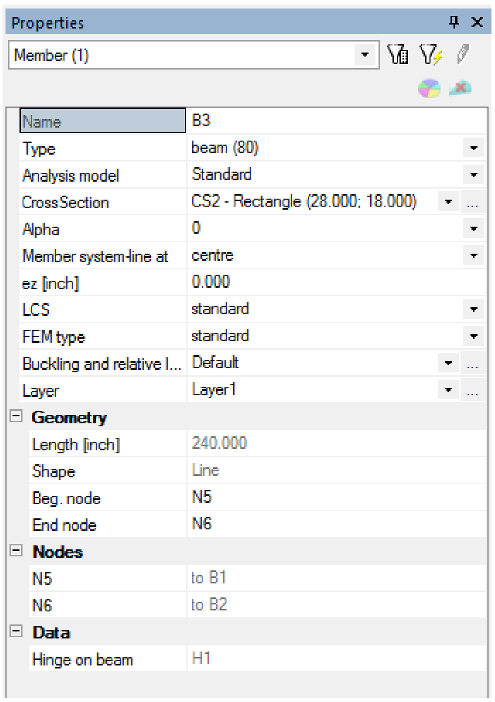
7. A window appears to indicate the number of connected nodes:



8. Connected nodes are represented in the graphical screen by means of double red lines:



If you select beam **B3**, the **Property Window** will show that node **N5** connects the beam to column **B1** and that node **N6** connects the beam to column **B2**.



Note:

*If a possible active selection is not deactivated when the **Connect members/nodes** command is used, the program will only search the nodes to be connected in this selection and not in the entire project.*

*It is also possible to perform the two previous operations at once. To do this you have to select the option **Check (merge duplicate nodes, erase invalid entities)** in the **Setup for connection of structural entities** dialogue box.*

9. Click **[Close]** at the bottom of the **Structure menu** to return to project view.

Graphical representation of the structure

Edit view

Within Scia Engineer there are several possibilities to edit the graphic representation of the construction. Below you will find the most important options:

- Edit the view point on the Construction
- Set a view direction
- Use the magnifier
- Edit view parameters through the menu **View parameters**

Editing the view point on the construction

One of the methods for editing the view point of the structure is through the three wheels in the bottom right of the graphic window; two are horizontal and one is vertical. With these **wheels** you can **zoom in** on the construction or **turn** it.

1. To zoom in on the construction or to turn the structure, click on the appropriate wheel (the cursor will change into a hand), keep the left mouse button pressed and move the wheel

   OR

   Set the view point by combining the buttons and mouse.

2. Press CTRL + right mouse button at the same time and move the mouse to **turn** the construction.

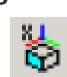
3. Press SHIFT + right mouse button at the same time and move the mouse **move** the construction.

4. Press CTRL + SHIFT + right mouse button at the same time and move the mouse to **zoom in or out** on the construction.

<u>*Note:*</u>

If the structure is being turned while a node is selected, the structure will turn around the selected node.

Setting a view direction with regard to the global coordinate system

1. Click on the button **View in direction- X**  for a view the structure in the X-direction.

2. Click on the button **View in direction- Y**  for a view the structure in the Y-direction.

3. Click on the button **View in direction- Z**  for a view the structure in the Z-direction.

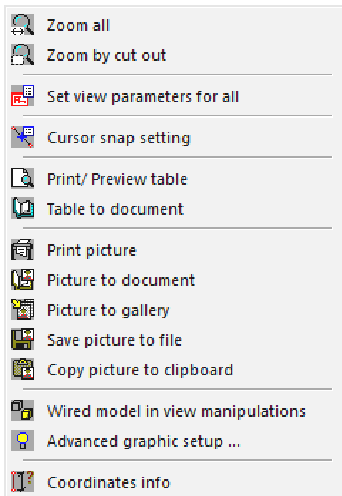
The magnifier

- Use  to enlarge the view size.
- Use  to decrease the view size.

- Use  to zoom in on a window.
- Use  to view the whole structure.
- Use  to zoom in on the selection.

Editing view parameters through the menu View parameters

1. Click in the graphic window on the right mouse button. The following shortcut menu appears:



Note:

If an element was selected previously, you can define a setting that only applies to the selected elements. (An adapted shortcut menu appears).

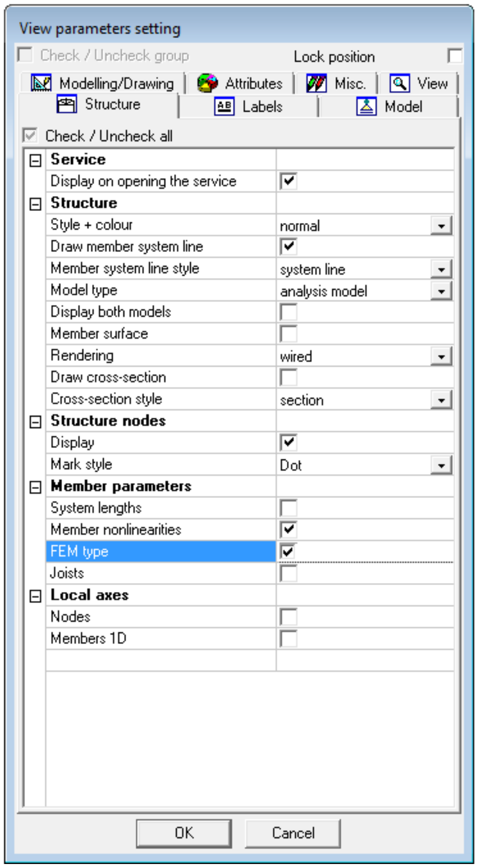
2. Choose the option **Set view parameters for all**. The window **View parameter setting** appears. The menu consists of various tabs. You can set the view parameters for all entities or just for the selected entities.

View parameters – Entities

Through the tab entities the representation of the different entities can be adapted.

In the group **Structure** the following items are important for this project:

- **Style and color:** You can display the colors per layer, material, cross-section or structural type.
- **Draw cross-section:** With this the symbol of the cross-section is displayed on every member.
- **Local axes:** With this the local axes of the elements are activated.



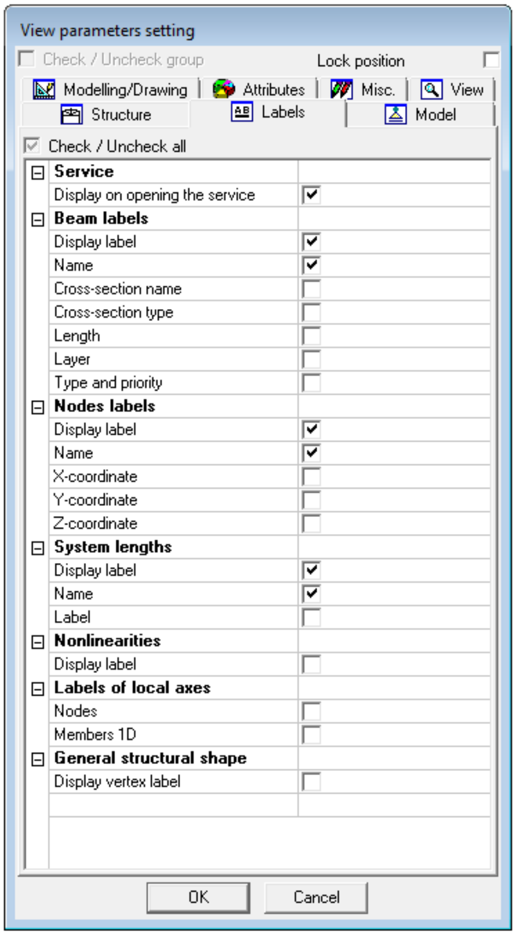
View parameters – Labels and description

Through the tab **Labels,** the labels of different entities can be displayed. In the group **Members** the following items can be displayed as a label on the structure:

- **Name:** Show the name of the cross-sections in the label.

- **Cross-section type**: Show the cross-section type in the label.

- **Length**: show the length of the member in the label.



View parameters – shortcuts

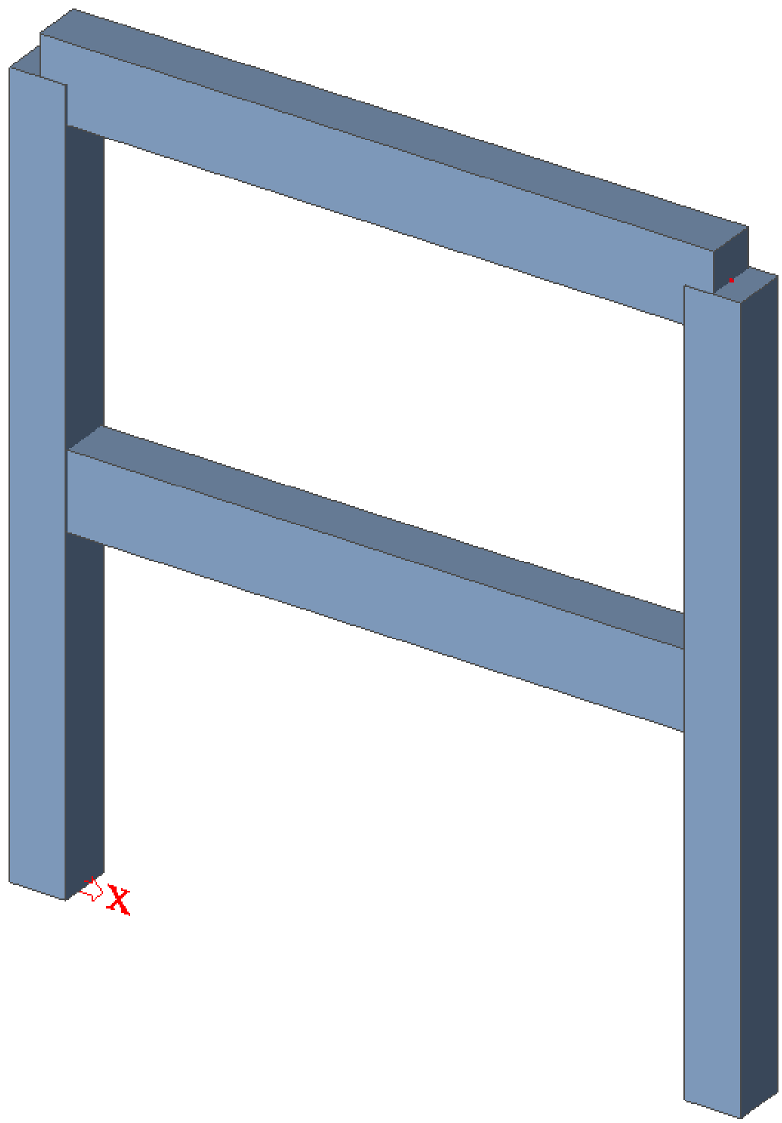
In the tool bar above the **Command line**, several frequently used options are grouped among which:

- **Show/hide surfaces**  to show the surfaces of the cross-sections.
- **Render geometry**  to view the rendered members in the structure.
- **Show/hide supports**  to show supports and hinges.
- **Show/hide load**  to show the load case.
- **Show/hide other model data**  to show other model data (like hinges, internal nodes, etc.)
- **Show/hide node labels**  to view the label of the nodes.
- **Show/hide member labels**  to view the label of members.

.

- **Set load case for view**  to edit the active load case.
- **Fast adjustment of view parameters on the whole construction**  to quickly access the options from the View Parameters menu.

After rendering, the following structure is obtained (AXO view):



Loads and combinations

Load Cases and Load Groups

Each load that is inserted into the project and added to the structure is attributed to a load case. A particular load case can contain many different load types.

Each load can is attributed properties which will determine the proper generation of load combinations. In addition, a specific load case will carry a specific action which can be set as permanent or variable.

In the case of variable load cases, each variable load has its own associated load group. The group contains information about the category of the load (service load, wind, snow, etc.) and its appearance (default, together, exclusive). In an exclusive group, the different loads attributed to the group cannot act together in a normal combination. Default combinations, on the other hand, will allow for simultaneous action of the loads in the same group within the load combination generator.

The way in which load cases are defined, is critical for the load combinations created by the generator. We recommend that you thoroughly read the chapter about loads and combinations in the reference manual found in the Help menu.

In this particular project, three load cases are entered:

- **LC1**: Permanent Load Case: Self weight of the concrete members
- **LC2**: Permanent Load Case: Weight of the Floor and Weight of the Roof (Superimposed Dead Load)
- **LC3**: Service Load Case: Service load on the Floor (Live Load)

Defining a Self Weight Load Case

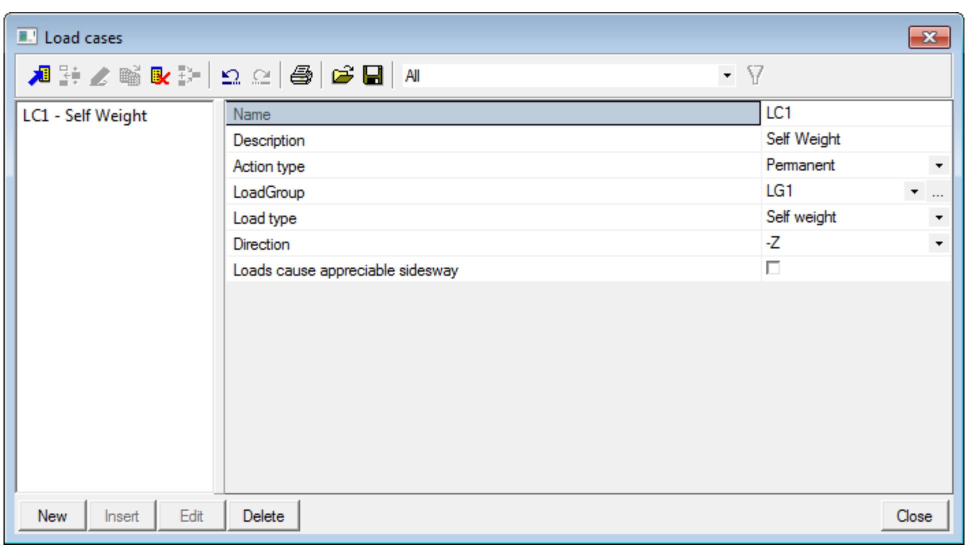
1. Double-click on 

in the **Main window**.

2. Before you can define loads, you first must enter load cases. Since this project does not contain any load cases yet, the **Load Cases Manager** will automatically appear.

3. By default, load case **LC1** is created. This load is a permanent load of the **Self Weight** load type. The self weight of the structure is automatically calculated by means of this type.

4. In the Description field, you can describe the content of this load case. For this project, enter the description **"Self Weight"**.



Defining a Permanent Load Case

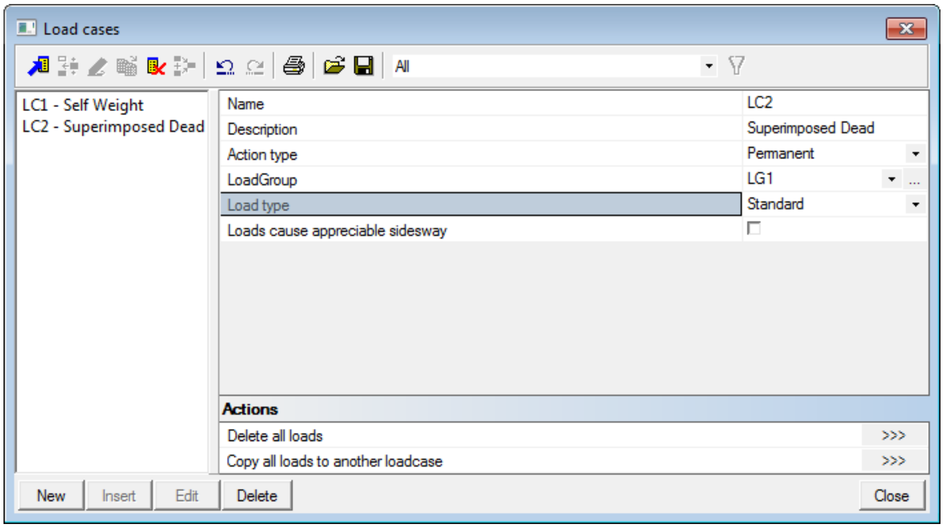
1. Click 

or  to create a second load case.

2. Enter the description **"Superimposed Dead"**.

3. As this is a permanent load, change the Action type to **Permanent**.

4. Verify that both LC1 – Self Weight and LC2 – Superimposed Dead are in the LG1 LoadGroup.

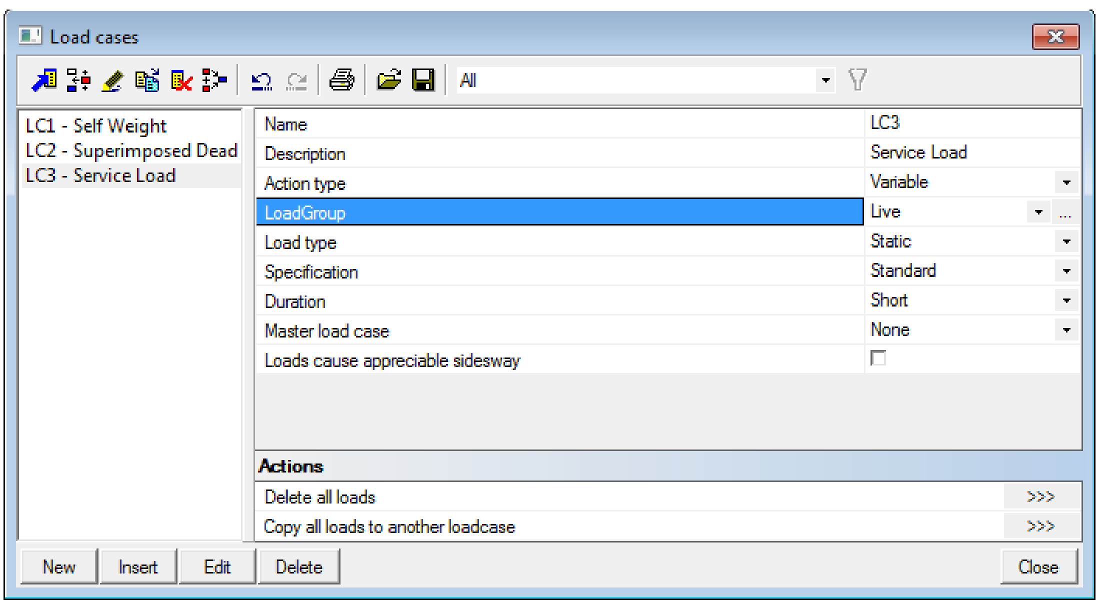


Defining a Variable Load Case

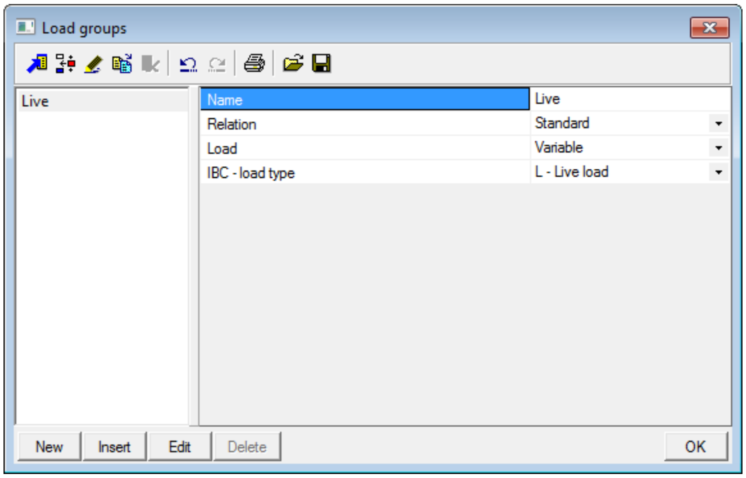
1. Click New or  to create a third load case.

2. Enter the description **"Service Load"**.

3. As this is a variable load, change the Action type to **Variable**.



4. The LoadGroup LG2 is automatically created. Click ... to display the properties of the LoadGroup.



The IBC load type determines the load factors that are attributed to the load cases in this load group. In this project, choose **L-live load.**

5. Click **[OK]** to close the **Load group manager** and to return to the **Load cases manager**.

6. Click **[Close]** to close the **Load cases manager**.

<u>Note:</u>

Load groups

Each load is classified in a group. These groups influence the combinations that are generated as well as the code-dependant factors to be applied. The following logic is adopted throughout the software:

Variable load cases that are independent from each other are associated to different variable groups. For each group, you set the load category (see ASCE 7*) and the combination factors from the American Concrete Code (ACI 318) are generated from the available load groups. When a generated combination contains two load cases belonging to different groups, reduction factors will be applied for the transient loads.*

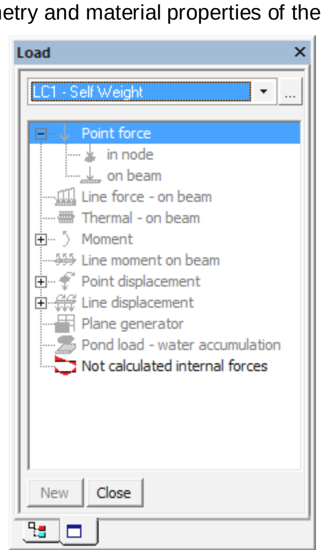
If the load is divisible, its different components are entered as individual load cases. As long as the load combination does not contain any variable load belonging to another group, no reduction factors will be applied. The different load cases of a divisible load are therefore associated to one variable group.

Load cases of the same type that may not act together, are put into one group, which is made exclusive, e.g. "Wind X" and "Wind -X" are associated to one exclusive group "Wind".

Loads

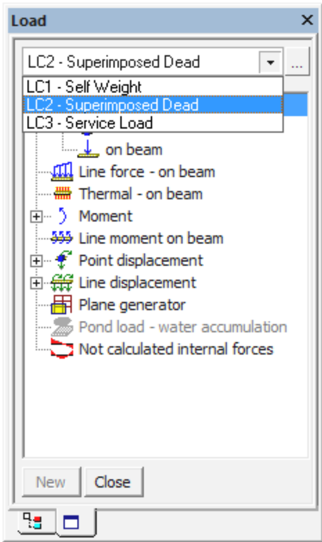
After the input of the Load cases, the **Loads menu** will automatically appear:

The first load case (**LC1**) includes the self weight of the concrete members. As can be seen in the Loads menu, self weight is automatically accounted for and no point or line loads need to be added to the frame. The self weight is added to the structure based on the geometry and material properties of the sections.



Switching between load cases

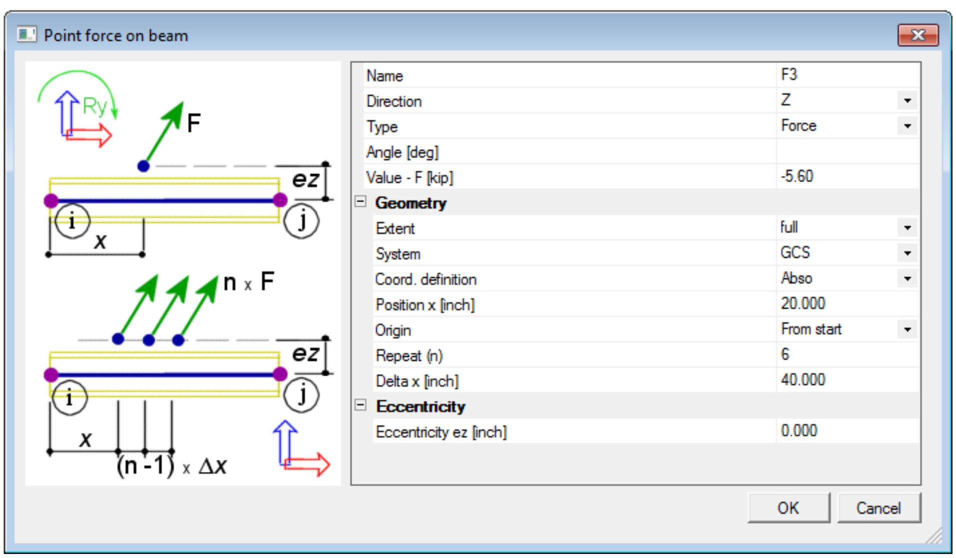
Activate LC2 by selecting this load case with the mouse pointer in the list box:



Once you have switched load cases, the floor and roof superimposed dead loads can be entered as a series of concentrated loads on the structure.

Entering a series of concentrated loads

1. Click on **Point force on beam** in the **Loads Menu**. The dialogue box **Point force on beam** appears.



2. The floor beam is loaded with 6 concentrated loads of **5.6 kips** with a spacing of **40 inches**. The first concentrated load in the series is at **20 inch** from begining node of the beam.

3. The Value of the concentrated load is changed to **–5.6 kips**.

4. The coordinate definition is set to **Absolute**.

5. The starting position, Position x is changed to **20 inch**.

6. The series consists of 6 concentrated loads, therefore set the Repeat (n) to **6**.

7. The spacing, Delta x between the concentrated loads is **40 inch**.

8. Confirm your input with **[OK]**.

9. Select beam **B3**.

10. Press **<Esc>** to terminate the input.

11. Press **<Esc>** once more to cancel the selection.

The roof beam is similarly loaded with concentrated loads of 2.8 kips. To complete this task, it is possible to copy the load from the floor beam to the roof beam and adapt it accordingly.

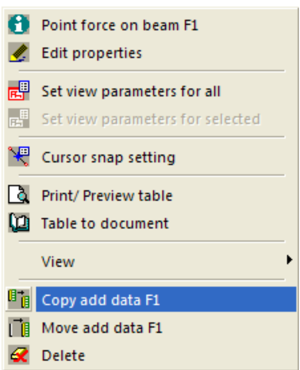
<u>*Note:*</u>

Loads, supports and hinges are considered as additional data, i.e. data that are additionally added to entities such as nodes and beams.

Copying loads

1. Select one of the concentrated loads on the floor beam with the left mouse button. As this concentrated load is part of a series, the entire series is automatically selected.

2. Press the right mouse button to display a popup menu:



3. Choose the option **Copy add data F1**.

4. Select the beam where this load should be copied to: the roof beam **B4**.

5. Press **<Esc>** to terminate the input.

6. Press **<Esc>** once more to cancel the selection.

Use the **Quick adaptation of view parameters on the entire construction** icon on top of the **Command line** to activate the **Labels of Loads** option in the **Loads/Masses** group. A normal load is represented in green.

Now, the value of the roof load can be changed.

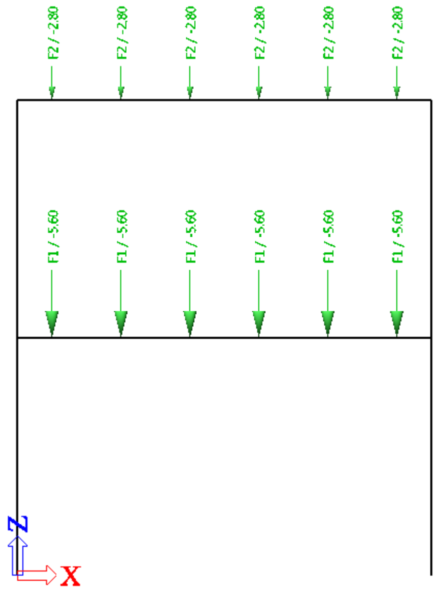
Adapting a load

1. Select one of the concentrated loads on the roof beam. As this concentrated load is part of a series, the entire series is automatically selected.

2. The properties of the series are displayed in the **Property Window**.

3. Change the value from **–5.6 kips** to **–2.8 kips**.

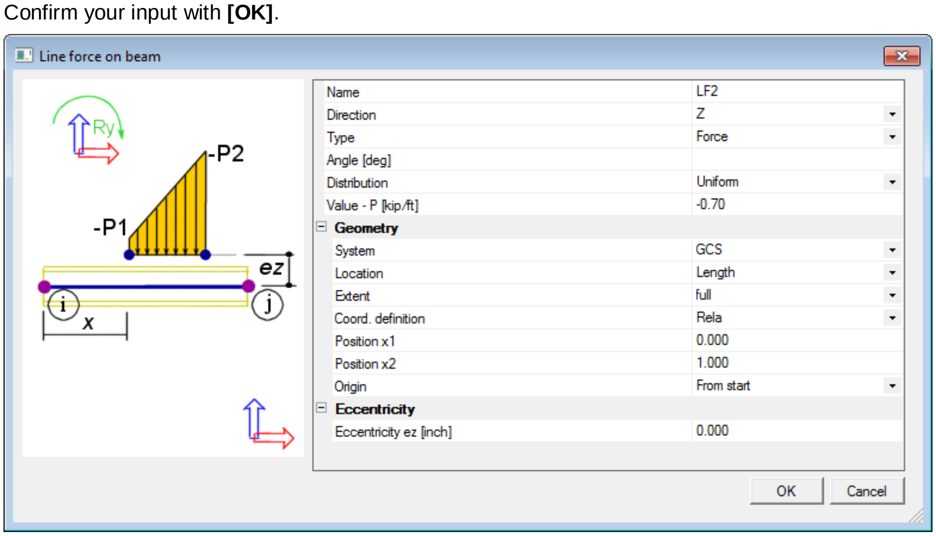
4. Confirm the change by pressing **<Enter>**.



After input of the loads in the LC2 load case, you can enter the service load, LC3 on the floor beam of -0.7 k/ft.

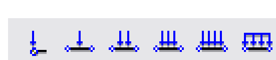
Entering a linear load

1. Click on **Line force – on beam** in the **Loads Menu**. The dialogue **Line force on beam** appears.
2. Change the type to **Force** and the value to -**0.7 kip/ft**.
3. Confirm your input with **[OK]**.



4. Select the beam on which this load must be positioned: floor beam **B3**.
5. Press **<Esc>** to terminate the input.
6. Press **<Esc>** once more to cancel the selection.
7. Click **[Close]** to leave the **Loads Menu** and to return to the **Main Window**.

Note:

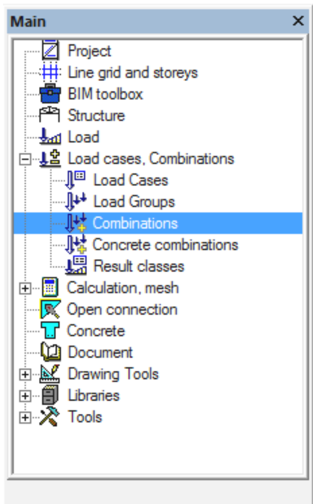
*The **Command line** includes a number of predefined loads:*  *which enable the fast and simple input of loads.*

Combinations

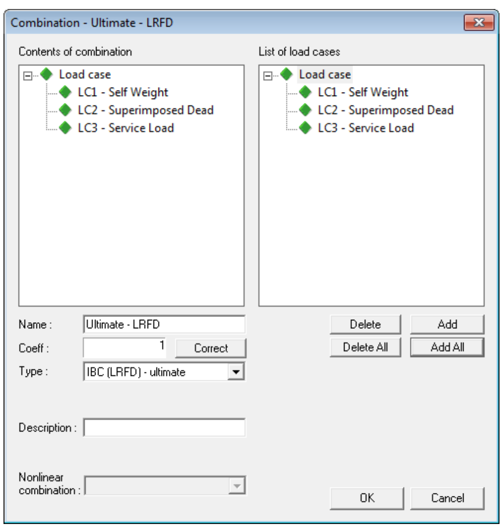
After input of the load cases, the latter can be grouped in combinations. In this project, two linear combinations are created, one for the Ultimate Limit State **(LRFD – Ultimate)** and one for the Ultimate Serviceability State **(LRFD – Service).**

Defining Combinations

1. Double-click on Combinations below Load cases, Combinations in the **Main window**.



2. Since no combination has been entered yet, the window to create a new combination will automatically appear.



3. The Type of the combination is changed to **IBC (LRFD) - ultimate**. With this combination type, Scia Engineer will automatically generate combinations in accordance with the complex composition rules of the ACI.

4. By means of the button **[Add all]**, all load cases can be added to the combination.

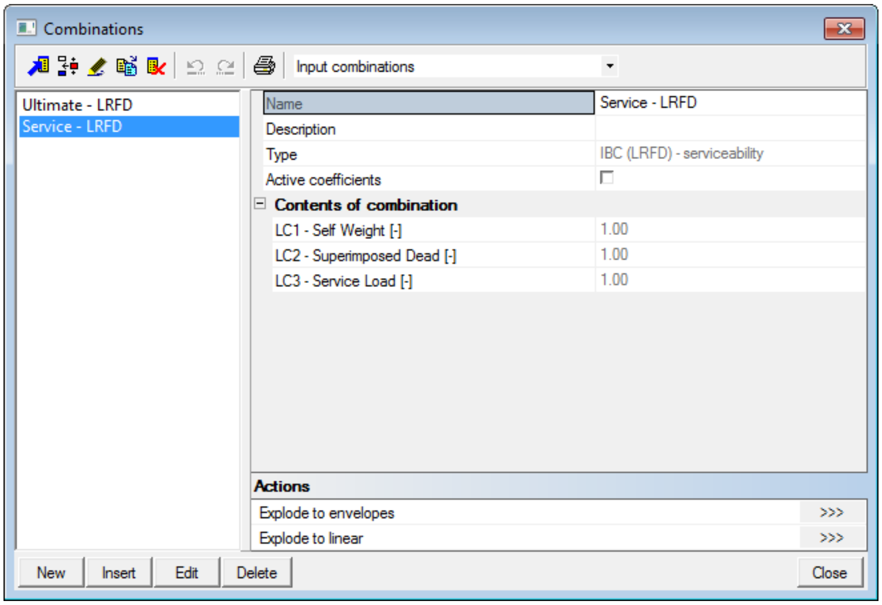
5. Confirm your input with **[OK]**. The **Combination Manager** is then opened.

6. Click New or to create a second combination.

7. Change the **Type** of the combination to **IBC (LRFD) - serviceability**.

8. Confirm your input with **[OK]**.

9. Click **[Close]** to close the **Combination manager**.


Combinations
Input combinations
Ultimate - LRFD
Service - LRFD
Name
Service - LRFD
Description
Type
IBC (LRFD) - serviceability
Active coefficients
Contents of combination
LC1 - Self Weight [-]
1.00
LC2 - Superimposed Dead [-]
1.00
LC3 - Service Load [-]
1.00
Actions
Explode to envelopes
>>>
Explode to linear
>>>
New
Insert
Edit
Delete
Close

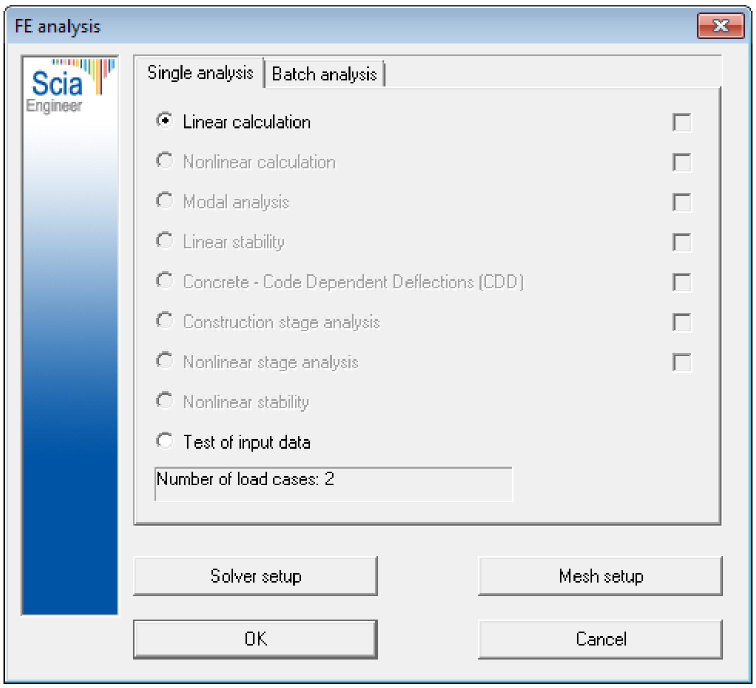
Calculation

Linear Calculation

If the calculation model is ready to be analyzed, it is then time to run the calculation of the frame and retrieve results.

Executing the Linear Calculation

1. Double-click on Calculation below Calculation,Mesh in the **Main window**.
2. The **FE Analysis** window appears. Click **[OK]** to start the calculation.



3. After the calculation, a window announces that the calculation is finished and the maximum deformation and rotation for the normative load case is shown. Click **[OK]** to close this window.

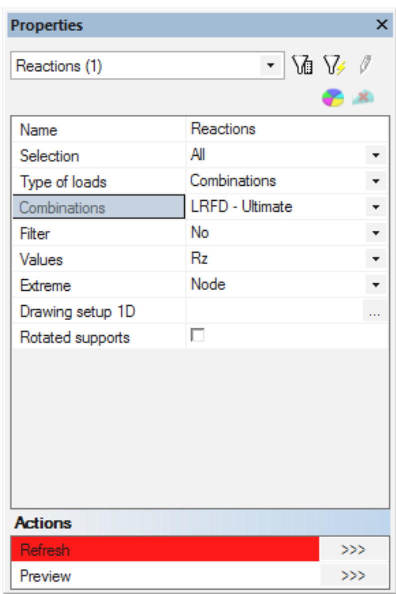
Results

Viewing results

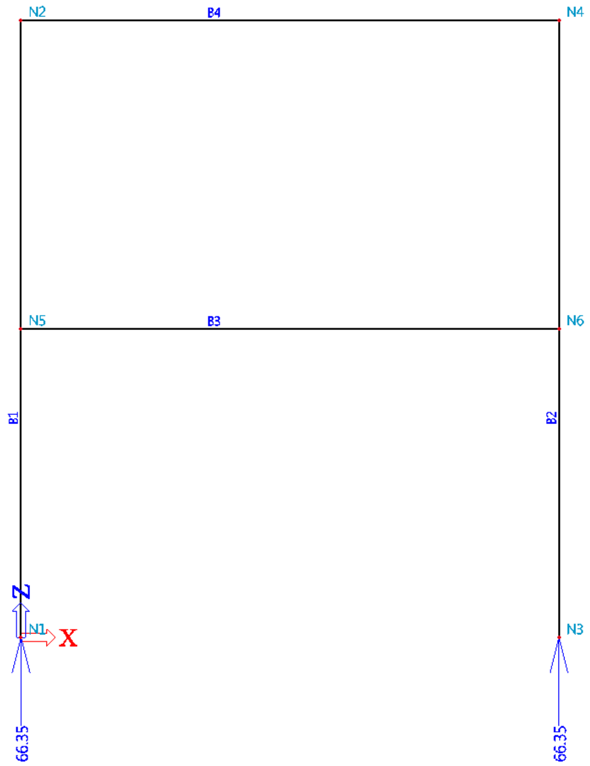
After the calculation is executed, the results of the loading on the frame can be viewed.

Viewing the Reaction Forces

1. Double-click on Results in the **Main window**. The **Results menu** then appears.
2. Below **Supports**, click **Reactions**.
3. The options in the **Property Window** are configured in the following way:
   - The Selection field is set to **All**.
   - The Load type is set to **Combinations** and the Combination to **ULS**.
   - The **Values** are wanted for **Rz**.
   - The **Extreme** field is changed to **Node**.



4. The action **Refresh** has a red background, i.e. the graphical screen must be refreshed. Click on the >>> button behind **Refresh** to display the results in the graphical screen in accordance with the set options.



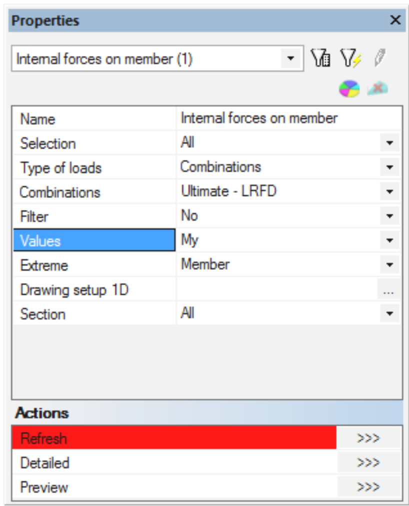
5. To display these results in a table, the **Preview** action is used. Click on the >>> behind **Preview** to open the Preview.

Note:

The Preview appears between the Graphical Screen and the Command line. This screen can be maximized to display more data at any time.

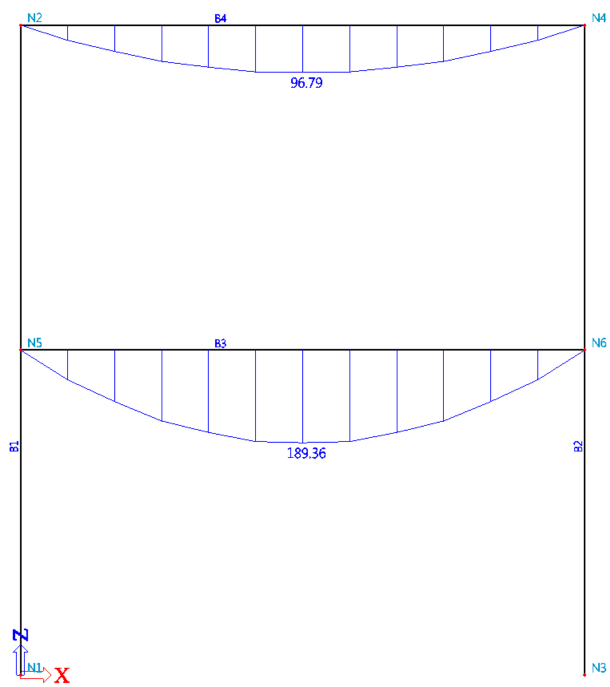
Viewing internal forces on beam

1. In the **Results** menu, open the **Beams** group and select **Internal forces on beams.**
2. The options in the **Property Window** are configured in the following way:
   - The Selection field is set to **Current**.
   - The Load type is set to **Combinations** and Combination to **ULS**
   - The Values are wanted for **My**.
   - The Extreme field is changed to **Member**.



3. Select the two beams **B3** and **B4** using the left mouse button.

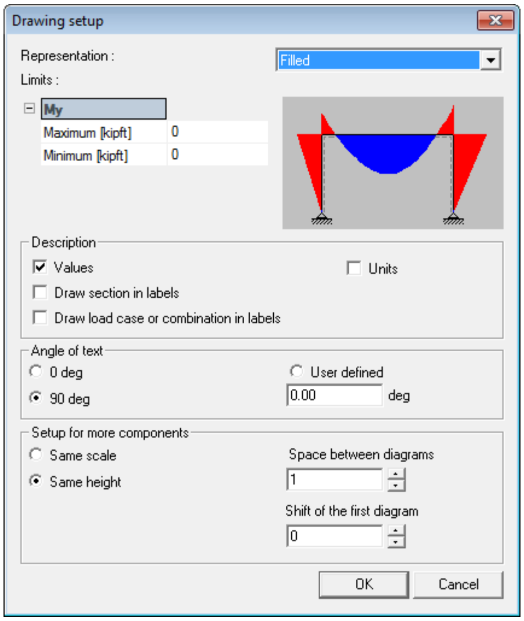
4. Click on the >>> button behind **Refresh** to display the results on the graphical screen in accordance with the set options.



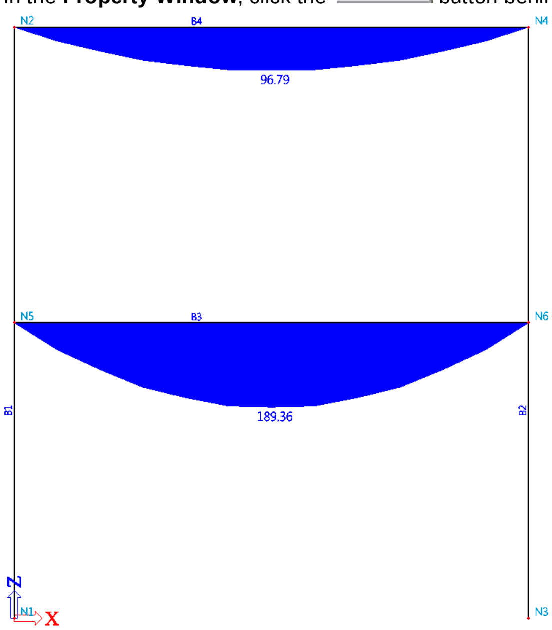
To change the display of the results, the settings of the Graphical Screen can be adapted.

Configuring the Graphical Screen

1. In the **Properties window**, click the ... icon behind **Drawing Setup**. The options for the graphical screen are opened.



2. In the **Representation** field, choose **Filled**.
3. The **Angle of text** is set to **0°**.
4. Click **[OK]** to confirm your input.
5. In the **Property Window**, click the >>> button behind **Refresh** to display the results in the graphical screen.



6. Then click **[Close]** to leave the **Results Menu**.
7. Press **<ESC>** to cancel the selection.

<u>Note:</u>

*To change the font size of the displayed results, you can use the **Setup > Fonts** menu. In this menu, the different sizes of the displayed labels can be changed as well as the sizes of any text associated with a specific layer.*

Code check

The concrete modules of Scia Engineer contain a number of powerful tools to execute concrete calculations in accordance with the chosen standard, in this case ACI 318-08.

The possibilities for calculations include:

- Input of advanced concrete data
- Calculation of the slenderness
- Reduction of M and V at the supports
- Design calculation of the Theoretically Required Reinforcement
- Input of the Practical Reinforcement

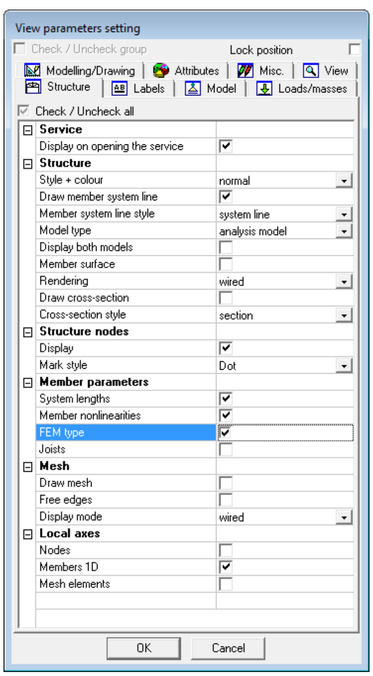
In this tutorial, we will only describe the basics of the concrete calculation. For more information regarding the advanced concrete calculations, please refer to the Advanced Training Curriculum.

Before you can start the concrete calculations, you first must check the buckling parameters of the beams. By means of the view parameters, the buckling lengths of the beams can be displayed on the structure.

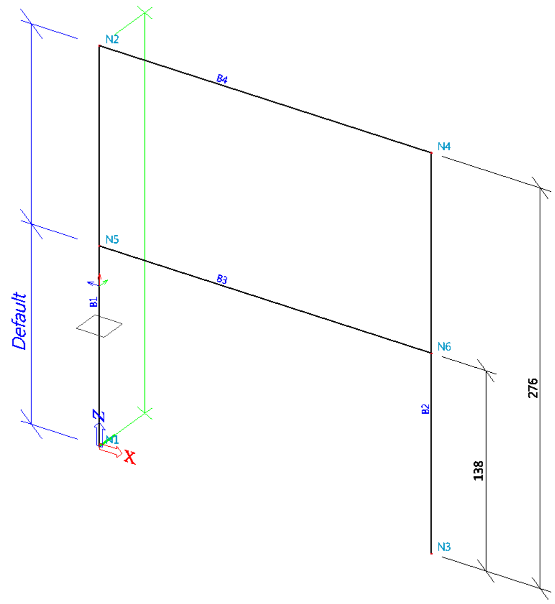
Buckling parameters

Displaying the system lengths

1. Using the left mouse button, select column **B1**, the left column of the frame.
2. Click the right mouse button at an arbitrary position in the workspace. A menu lists the possibilities for the selected entity.
3. In this menu, select the Set view parameters for selected option. The **View parameter settings** window appears.



4. Activate the **System lengths** and **Draw cross-section** options to display the reference lengths and the cross section shape of the beam.
5. Activate the **Local axes – Members 1D** option to display the local coordinate system of the beam.
6. Confirm your input with **[OK]**.
7. Press **<ESC>** to cancel the selection.



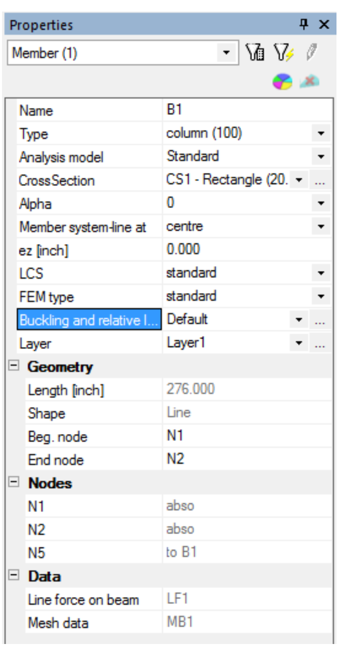
The figure shows that system length ly for buckling around the strong axis (y-y) is 138 inch while the lz for buckling around the weak axis (z-z) is 276 inch. The beam in the middle of the column therefore supports the column for buckling around the strong axis.

To modify the buckling data of a member, use the option **Buckling and relative lengths** in the **Property window** of the selected member to change the members buckling parameters and relative length.

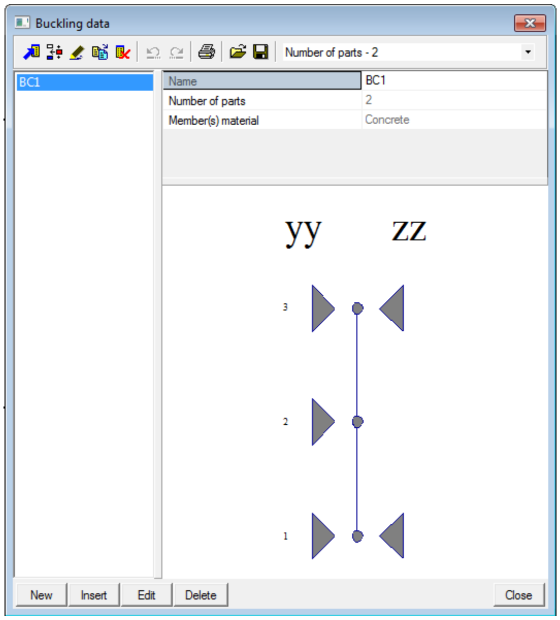
Setting the Buckling Parameters

1. Select both columns with the left mouse button.

2. The **Properties window** shows the common properties of both entities. The **Buckling and relative lengths** are set to **Default**.

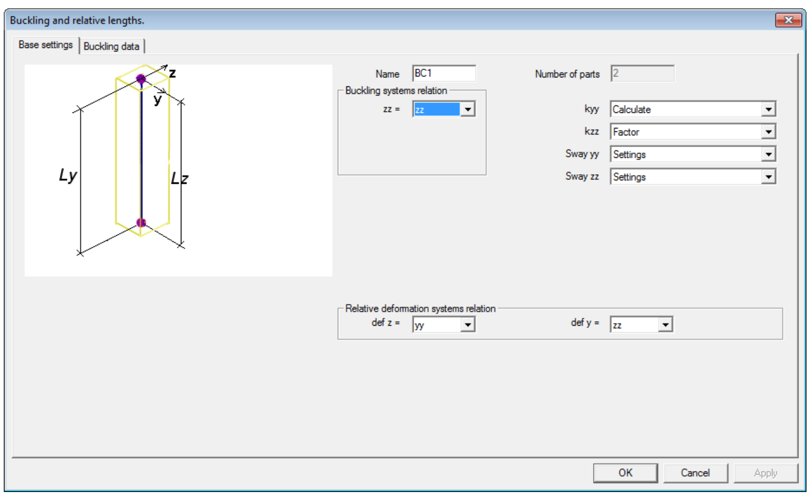


3. Click the ... icon behind **Buckling and relative lengths**. The **Buckling data** window appears.



This window shows that the column is supported in the middle for buckling around the strong axis (Y-Y) but not for buckling around the weak axis (Z-Z).

4. Click **[Edit]** to change the buckling data. The **Buckling and relative lengths** window appears.



5. On the **Base Settings** tab, several data can be changed.

- The **Name** field contains the name of the buckling parameter, in this case **BC1**.

- **kyy** and **kzz**: in these fields, you can indicate if the program must calculate the buckling factor around the axis regarded or if you prefer entering this factor manually. A third option allows for a manual input of the buckling length. The Support option can be used to determine the buckling factor in accordance with the applicable cross section and applicable material code.

- **zz**: in this field, you can indicate the system length to be used for the weak axis.

- **Sway yy** and **Sway zz**: in these fields, you can indicate if the member is braced or not in the direction regarded. When you choose the Settings option, the default settings are used.

<u>*Note:*</u>

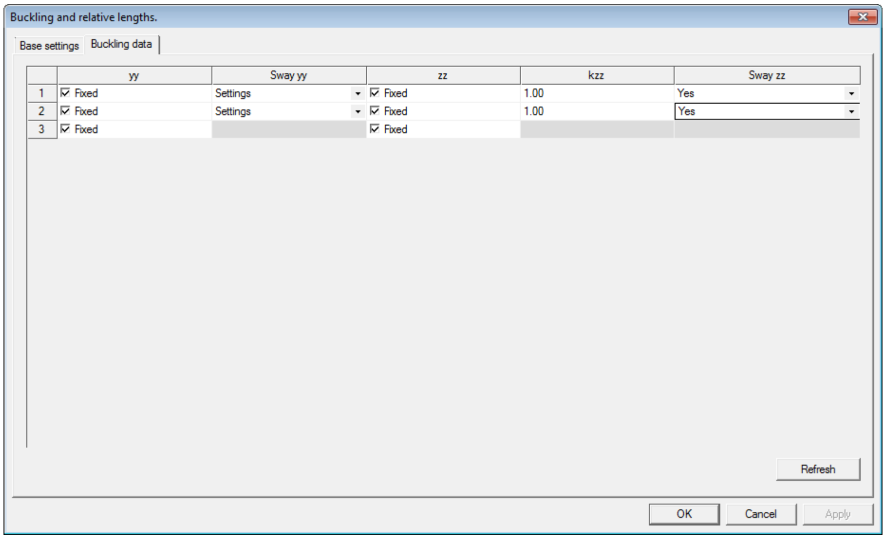
*The default settings for the buckling parameters are displayed in **Concrete > 1D member >** **Setup > Design defaults**. By default, both directions are unbraced for a concrete calculation. So for example, in a concrete calculation, no wind bracings are assumed.*

- **def z** and **def y**: in these fields, you can indicate the system length to be used for the relative deformations.

6. On the **Buckling data** tab, you can change the parameters in detail. As the columns consist of 2 components, 3 positions are available: (1) at the beginning, (2) in the center at the floor beam and (3) at the end.

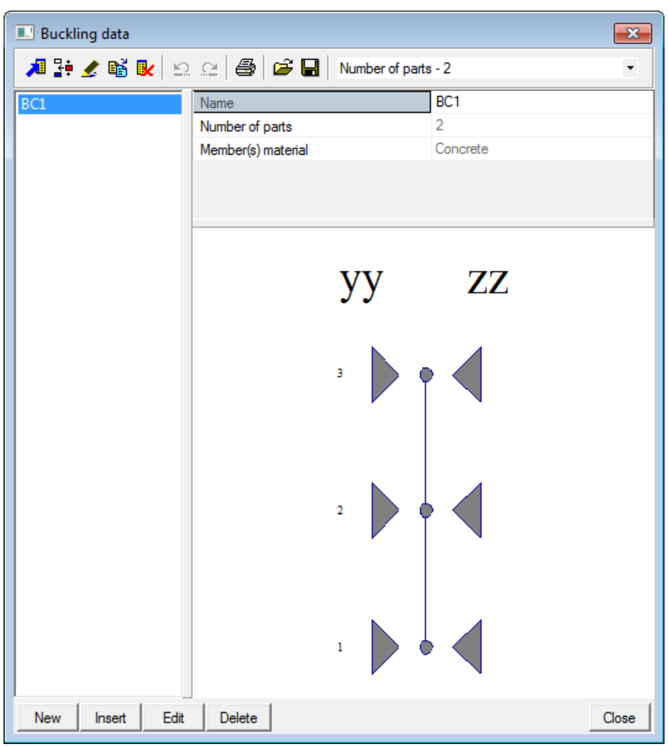
   For this project, we assume that the columns are not braced in both directions. Therefore, the **Non-braced** property can be set to **Yes** both for the strong and the weak axis. This could also be left on settings, since default settings are sway for concrete calculation.

   For this project, we also assume that the column is supported in the middle for buckling around the weak axis. The **zz** property at position (2) therefore can be set to **Fixed**.



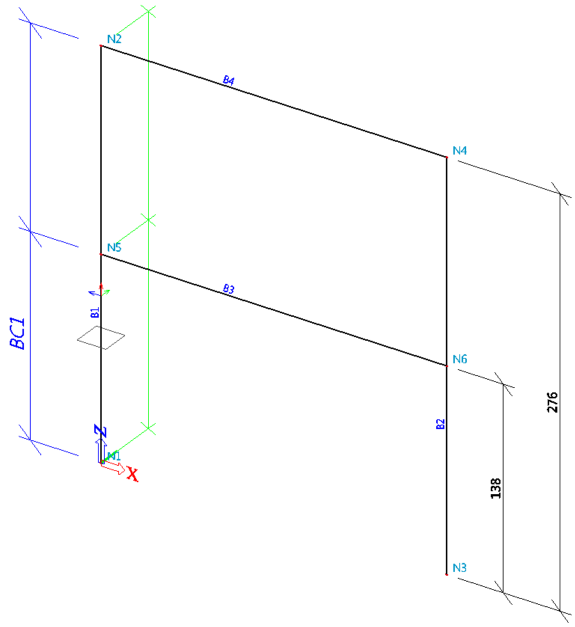
7. Click **[OK]** to close this window.

8. The **Buckling data** window re-appears and displays the changed buckling data. Click **[Close]** to close this window.



9. The **Properties window** indicates that the buckling parameter **BC1** is used for the columns in the concrete frame.

10. Press **<Esc>** to cancel the selection.



When the buckling parameters are set, you can continue with the concrete calculations. Before proceeding, deactivate the representation of the View parameters through Quick adaptation of view parameters on the entire construction.

When the buckling data is adapted, you must recalculate the project. For these steps, refer to the section of the tutorial concerning the **Linear calculation.**

Concrete calculation

Double-click on Concrete in the **Main window** to open the **Concrete menu**.

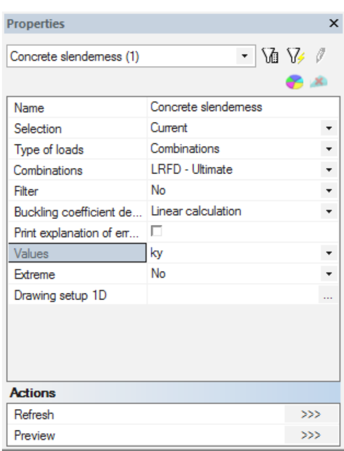
Displaying the Slenderness and the Buckling Lengths

1. Click the Concrete slenderness icon under the **1D member tree**

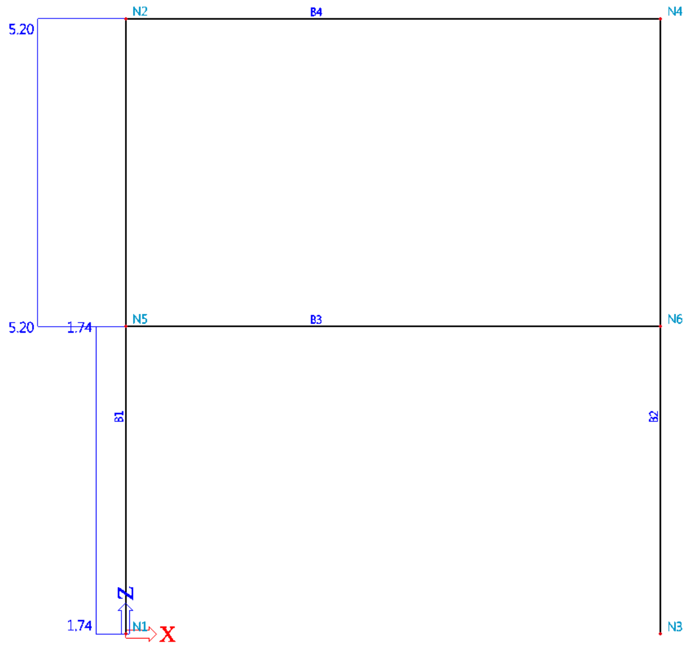
2. The options in the **Properties window** are configured in the following way:

   - The **Selection** field is set to **Current**.
   - The Load type is set to **Combinations** and Combination to **LRFD - Ultimate**.
   - The **Values** are desired for **ky**, i.e. the slenderness around the yy axis.
   - The **Extreme** field is modified to **No**.

3. Select column **B1**, the left column of the frame.

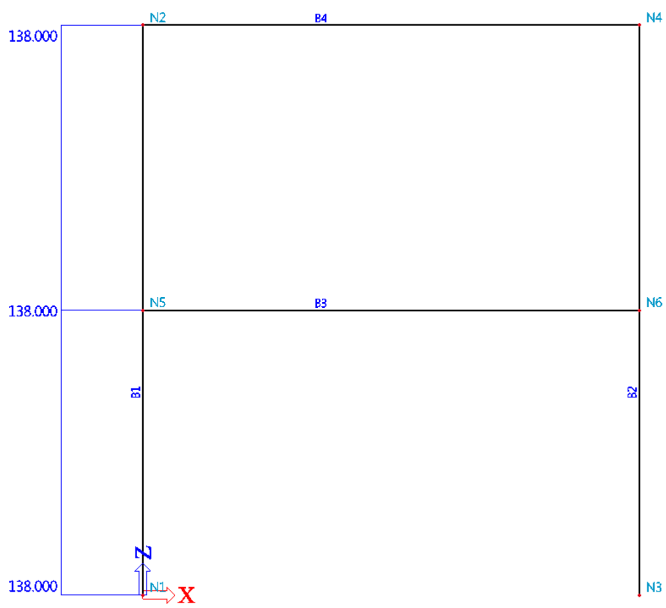


4. In the **Property Window**, click the >>> button behind **Refresh** to display the results in the graphical screen in accordance with the set options.



5. Change the **Values** field to **luy** to display the reference length for buckling around the strong axis.

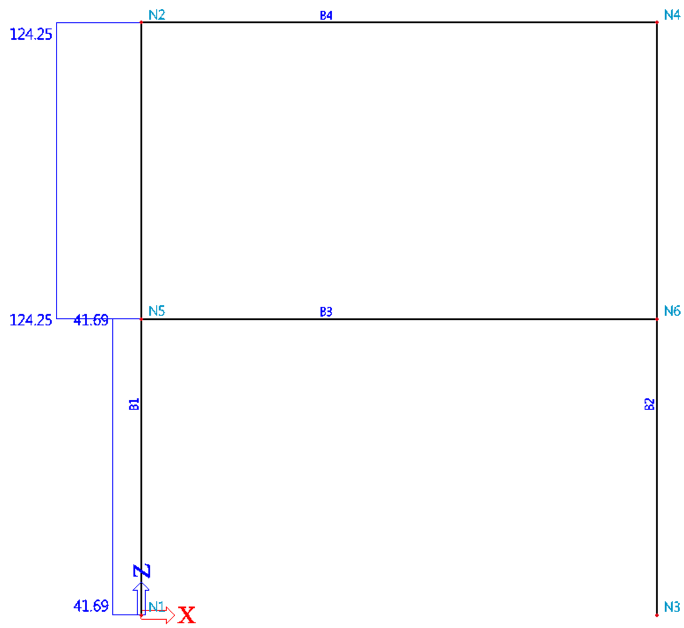
6. In the **Property Window**, click the >>> button behind **Refresh** to display the new set values.



As already indicated in the buckling parameters, the reference length is **138 inches.**

7. Change the **Values** field to **ky*luy/ry** to display the buckling length for buckling around the strong axis.

8. In the **Property Window**, click the >>> button behind **Refresh** to display the new set value.



Note:

The calculation of slenderness is important for reinforcement calculation in columns. It will determine whether a second order moment has to be taken into account. For a detailed explanation of this, refer to the Advance Concrete Training tutorials.

Theoretically required reinforcement

The design of theoretical reinforcement can be found under Member design - Design in the concrete menu.

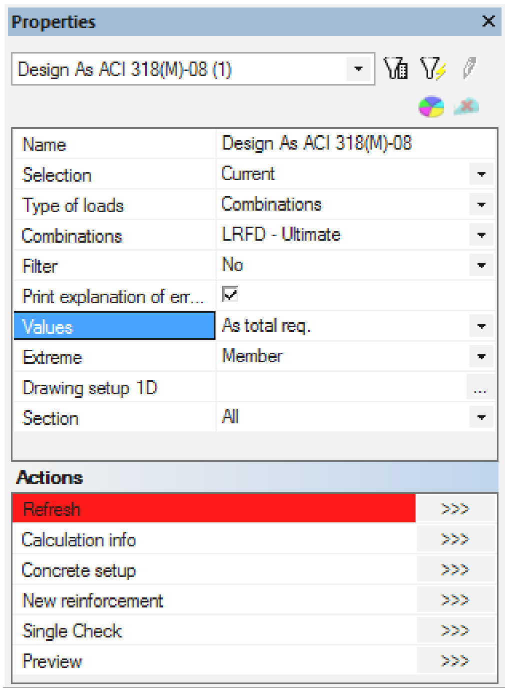
In this tutorial the reinforcement calculation will only be completed and illustrated for the beam B3. However, it is understood that the reinforcement calculation for the other members is similar. For background on the ACI reinforcement calculations, refer to the document, Design of Reinforcement for ACI 318.

Note:

The calculation procedure for columns is different than for beams. Beams will produce an output for upper and lower reinforcement. Column reinforcement is detailing as being on one or both sides, depending on the calculation type which can be uni-axial or bi-axial bending. This theory goes beyond the scope of this tutorial, for more information, reference the Design of Reinforcement for ACI 318.

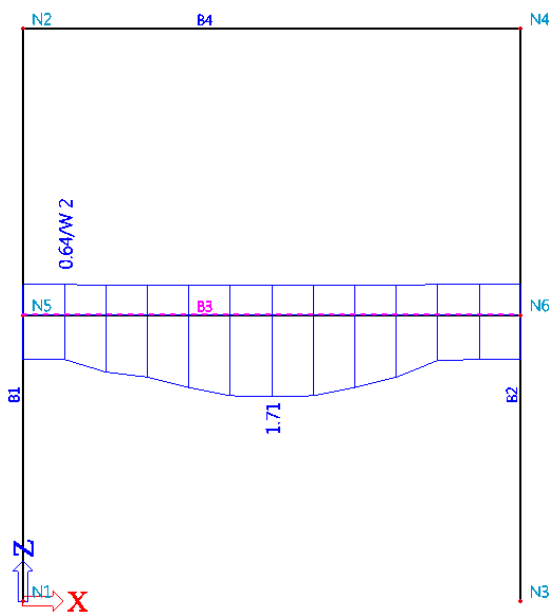
Longitudinal reinforcement As

1. In the **concrete menu**, go to Member design - Design.
2. Press **<Esc>** to cancel the selection.
3. The options in the **Property Window** are configured in the following way:
    - The Selection field is set to **Current**.
    - The Type of load is set to **Combinations** and Combination to **LRFD - Ultimate**.
    - The Values are wanted for **As total req**.
    - The Extreme field is changed to **Member**.



4. Select the beam **B3** with the left mouse button.

5. Click the >>> button behind **Refresh** to display the results on the graphical screen in accordance with the set options.



6. Click the >>> button behind **Preview** to display the results on the graphical screen in accordance with the set options.

Main lower reinforcement for selected beams

Member	d_x [inch]	Case	Φ [-]	N_u [kip]	M_{uy} [kipft]	M_{uz} [kipft]	c [inch]	d [inch]	$A_{s,add}$ [inch²]	$A_{s,user}$ [inch²]	Reinf.[no.]
B3	120.000	LRFD - Ultimate/2	0.90	0.00	189.36	0.00	2.854	25.750	0.82	0.88	2x0.750+2x0.750(1.77)

7. Click the >>> button behind **Single Check** to display the calculation detailed information.

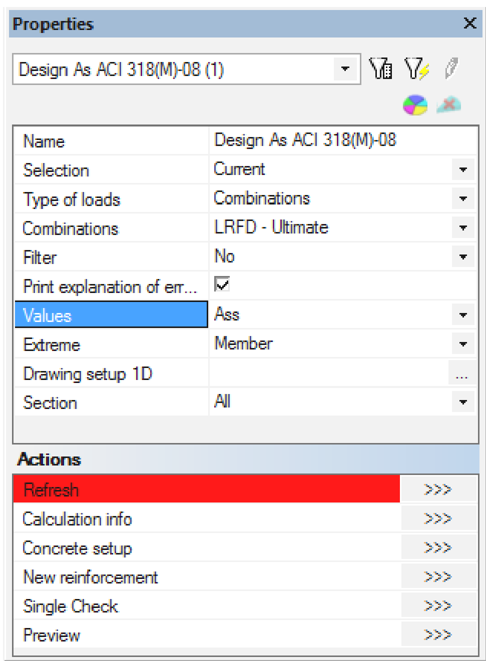
Note:

- *As,req is the theoretically required reinforcement.*
- *Reinf.[no.] represents the reinforcement proposal by Scia Engineer (based on the default diameter set in concrete setup)*
- *W/E: these are warnings and errors which serve as a guideline to give extra information to the user.*

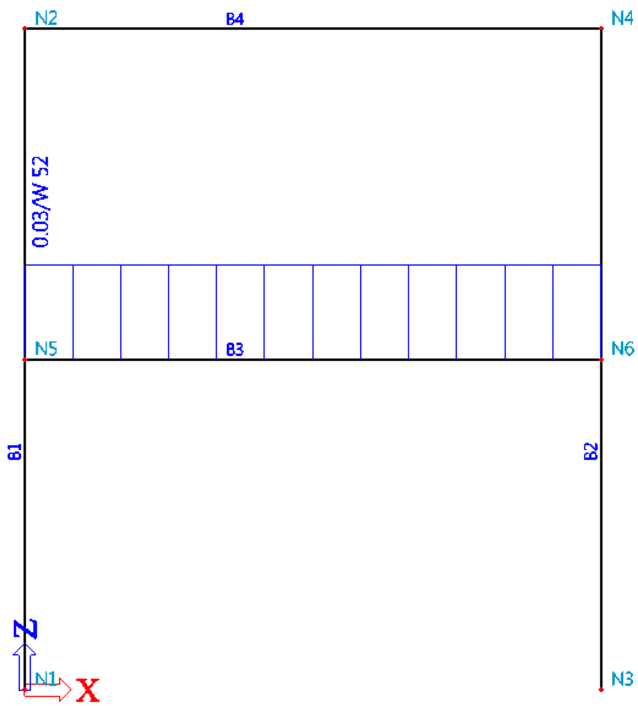
  *It is possible during the calculation that also warnings and/or errors appear. If you tick on **Calculation info** in the property window, an explanation of the applicable errors or warnings appear in the preview. The aim of this errors and warnings is to give extra information to the user at this point. The most common warnings are about the distance between bars which is too big or too small according to the detailing provisions and code.*

Transverse reinforcement Ass

1. In the **concrete menu**, go to Member design - Design.
2. Press **<Esc>** to cancel the selection.
3. The options in the **Property Window** are configured in the following way:
   - The Selection field is set to **Current**.
   - The Type of load is set to **Combinations** and Combination to **LRFD - Ultimate**.
   - Tick on Print explanation of errors and warnings
   - The Values are wanted for **Ass**.
   - The Extreme field is changed to **Section**



4. Select the beam **B3** with the left mouse button.
5. Click the >>> button behind **Refresh** to display the results on the graphical screen in accordance with the set options.



6. Click the >>> button behind **Preview** to display the results on the graphical screen in accordance with the set options.

Shear reinforcement for selected members

Member	d_x [inch]	Case	V_u [kip]	b_w [inch]	d [inch]	A_v [$inch^2$/inch]	Reinf.[no.]	W/E
B4	0.000	Ultimate - LRFD/3	37.87	18.000	25.750	0.03	3x0.375-12.875	52

Explanation of warnings and errors

52	The shear reinforcement was designed according to code longitudinal distance of stirrups.

7. Click the >>> button behind **Single Check** to display the calculation detailed information.

Member Data

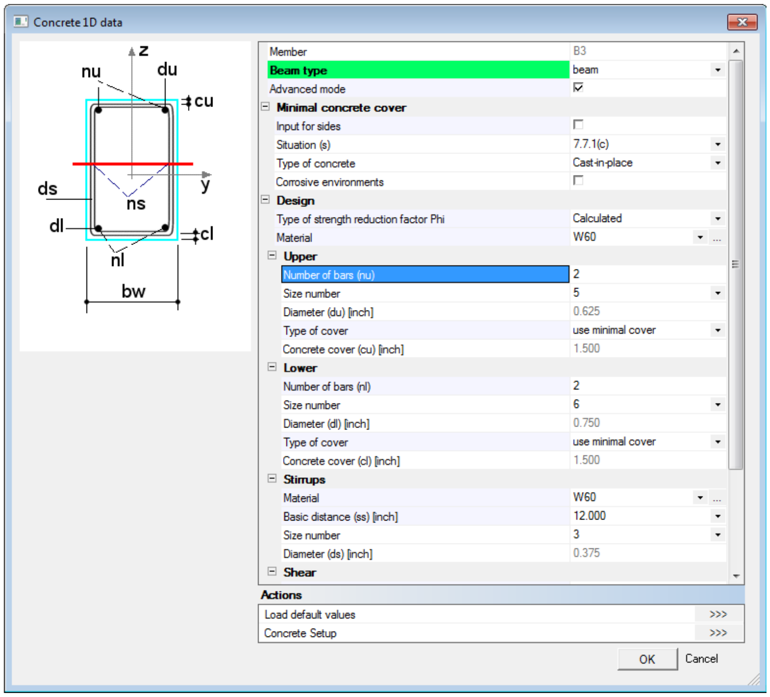
Member data can be used to change the settings for each member separately. The settings of the member data have a higher priority than the general concrete setup. For instance if you want to change the default diameter for each beam separately, member data could be used.

<u>*Note:*</u>

It should be noted that even for the calculation of theoretical reinforcement the default diameter is important. This is because it determines the lever arm d which is used for the calculation of As, required.

Member data can also be used to enter basic user reinforcement for the member.

1. Double-click Member data in the **Concrete Menu**.
2. Select floor beam **B3**.
3. The window **Concrete 1D Data** appears.



4. In this window, the concrete properties of the beam can be changed. For example you can modify the bar diameters, concrete coverage, reinforcement material, number of bars and stirrup size and spacing. In addition, you can define the basic user reinforcement for the beam by entering the number of longitudinal bars.

   For this example, enter **two #5 bars (.625" DIA)** for the upper longitudinal reinforcement and **two #6 bars (.75" DIA)** for the lower longitudinal reinforcement. Additionally, use **#3 stirrups (.375" DIA)** at a longitudinal distance of **12"** as shear reinforcement.

5. Click **[OK]** to confirm your input. The concrete data is added to the selected bar.

6. Press **<Esc>** to terminate the input.

7. Press **<Esc>** once more to cancel the selection.

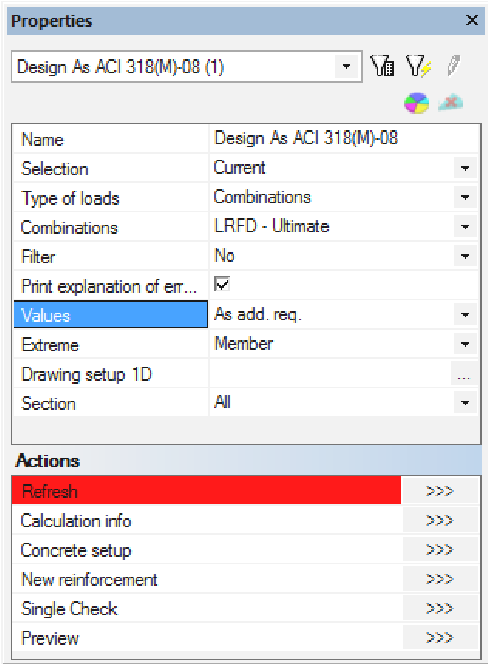
Additional reinforcement

If basic user reinforcement is input on a member, then it is possible to calculate how much reinforcement is additionally required.

1. In the **concrete menu**, go to Member design - Design.

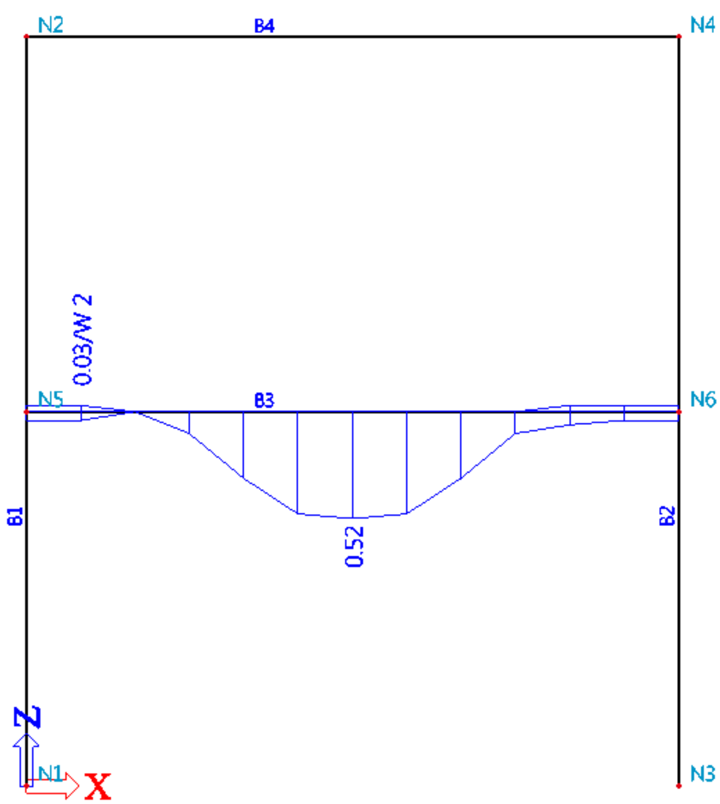
2. Press **<Esc>** to cancel the selection.

3. The options in the **Property Window** are configured in the following way:
   - The Selection field is set to **Current**.
   - The Type of load is set to **Combinations** and Combination to **LRFD - Ultimate**.
   - The Values are wanted for **As add.req**.
   - The Extreme field is changed to **Member**.



4. Select the beam B3 with the left mouse button.

5. Click the >>> button behind **Refresh** to display the results on the graphical screen in accordance with the set options.



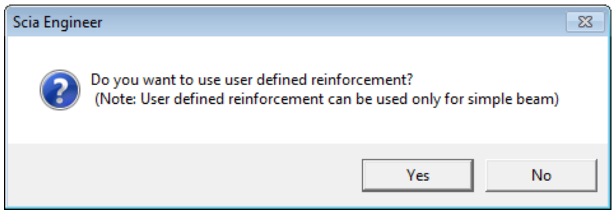
Practical reinforcement

1. Press **<Esc>** to cancel the selection.

2. To enter the practical reinforcement desired, you can open the group Redes (without As) and next select the item New reinforcement

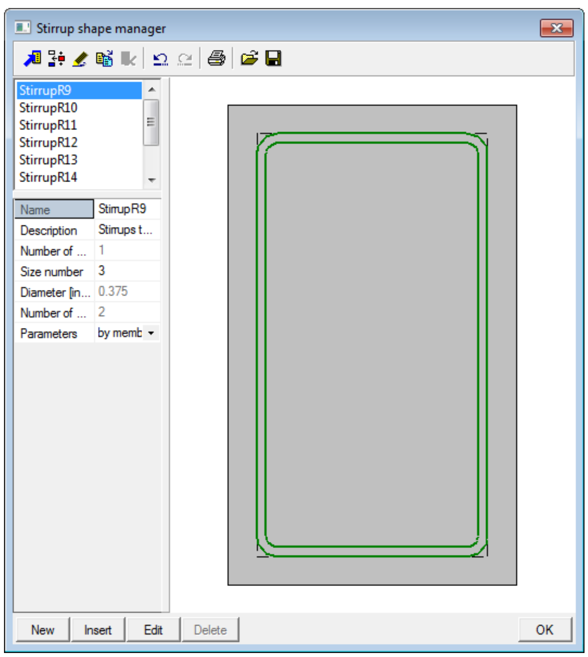
3. Alternatively, you can use the shortcuts in the command line. Click the Add reinforcement on whole beam icon.

4. Select floor beam B3 with the left mouse button and when doing so a message box will appear to ask if the basic user reinforcement of the concrete beam should be converted into practical reinforcement.

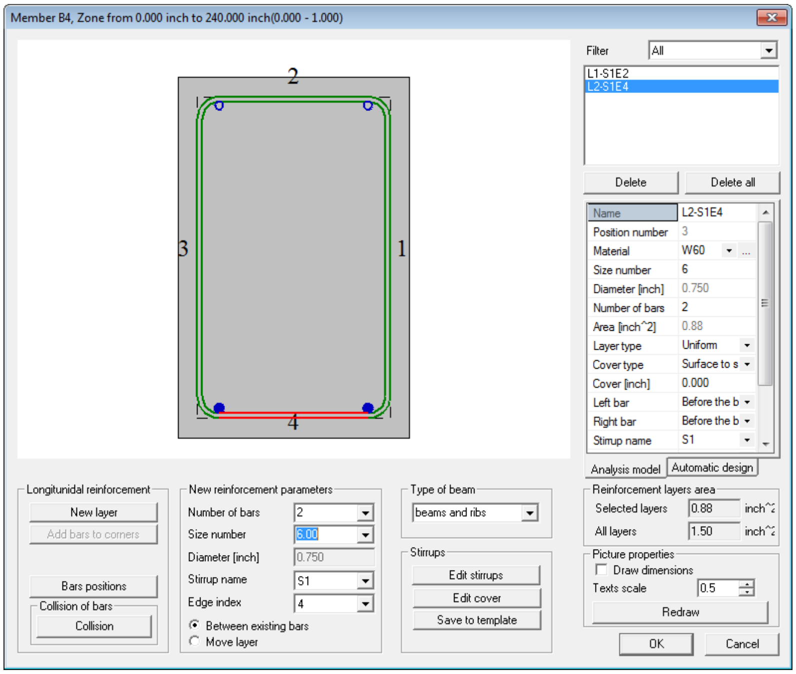


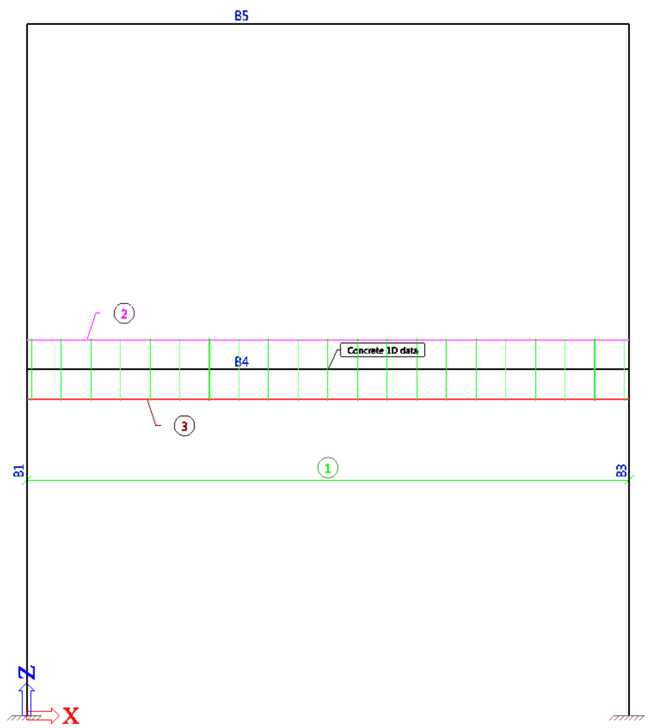
5. Click **[Yes]** to accept and proceed.

6. The stirrup shape manager will appear. In this window, choose the default **StirrupR9** and click **[OK]**.



7. The next screen shows two layers of longitudinal reinforcement that are included in the cross section. The upper layer consists of 2 #5 bars (.625” DIA) and the bottom layer consists of 2 #6 bars (.75” DIA). Accept these specifications for the cross section by clicking **[OK].**





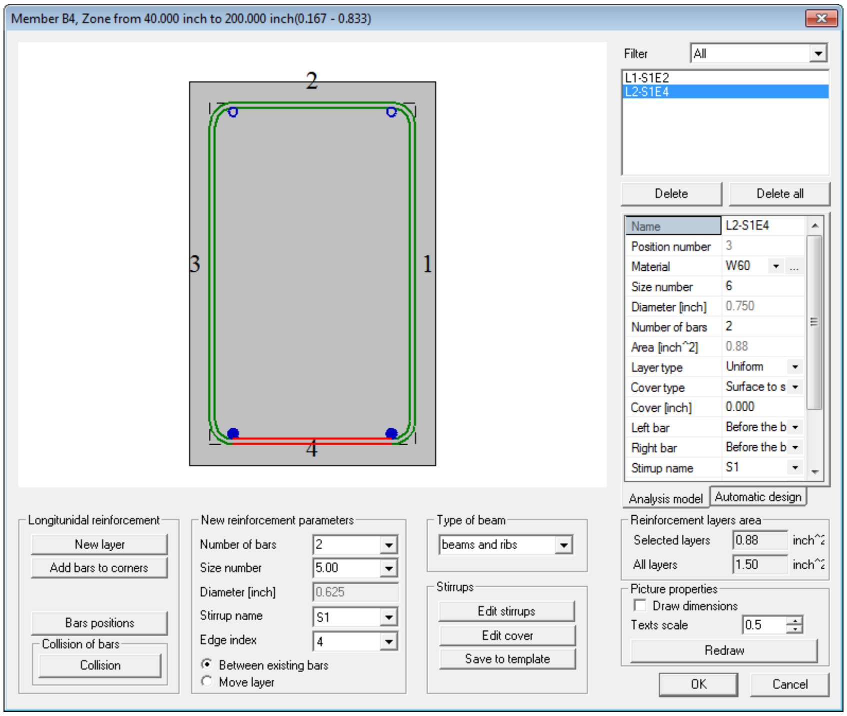
8. In previous steps we had calculated that some additional reinforcement was required in the middle of the concrete beam. Therefore it is necessary that we add a layer of reinforcement in the zone of the beam from 3'-4" to 16'-8".

To enter an additional layer of reinforcement on a selected interval you can either use the New longitudinal reinforcement icon which can be found in the concrete menu under Redes (without As) or use the shortcut in the command line to add longitudinal reinforcement on a selected interval .

To add longitudinal reinforcement on a selected interval using the shortcut, first select the icon and then select the beam that you want to add the reinforcement too (Beam B3 in this case). After the beam is selected you have to define the interval.

To define the interval it is helpful to use snap points on the beam. To do this, select the cursor snap settings icon in the command line. In the cursor snap settings menu activate option **h)** to create **6** snap points on each beam. Once you do this you will now have a snap point at 3'-4" increments along beam B3, now you can select an interval from 3'-4" to 16'-8" along beam B3.

9. The next screen shows us the current reinforcement layers in the zone from 3'-4" to 16'-8" along beam B3.

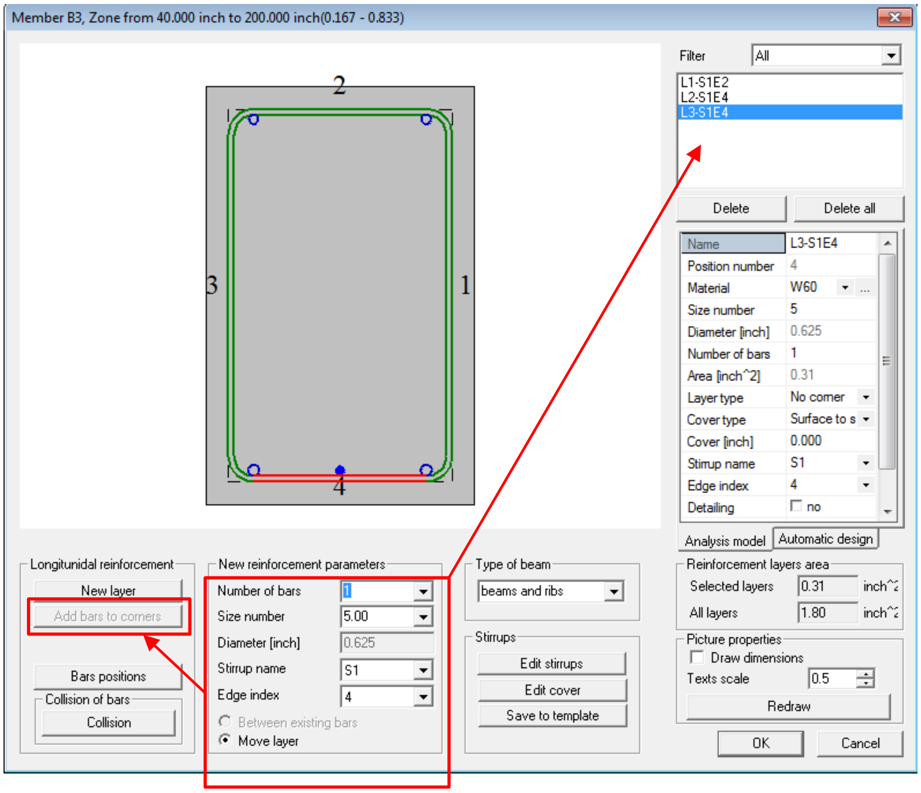


10. In this screen you can create new layers of reinforcement by using the icons in the bottom left corner of the menu. In this example input the following under New reinforcement parameters:

- Number of bars: **1**
- Size number: **5**
- Stirrup name: **S1**
- Edge index: **4**

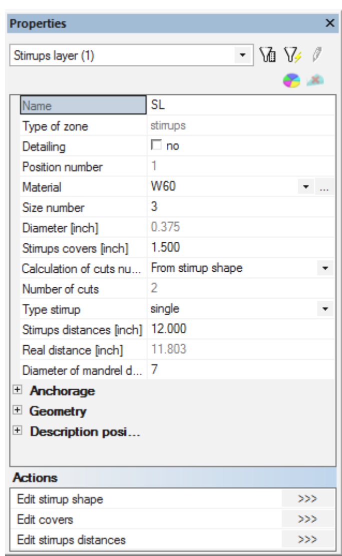
Once these new parameters are set, click on the 

icon to create the new reinforcement layer.



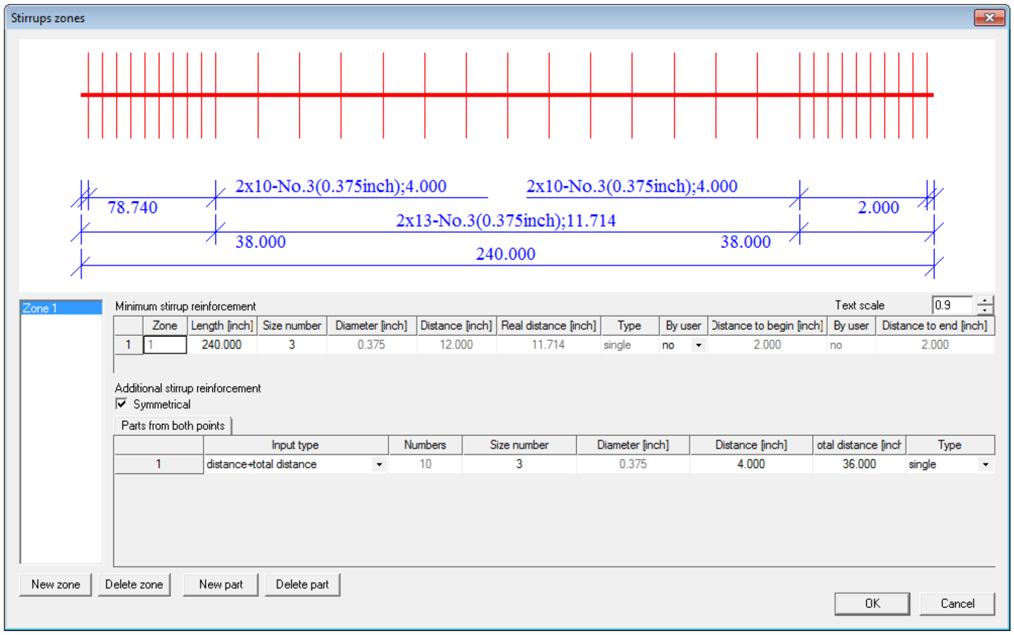
Once input, practical reinforcement layers are considered additional data. Since they are additional data, they can be easily copied to other beams. In addition, a reinforcement layers property can be easily viewed and modified by selecting the layer to open the property window.

11. We can now select the layer which contains the stirrups to modify the stirrup spacing.
    a. Select the stirrup **Reinforcement Layer** by clicking with the left mouse button on the circled number 1 1 .
    b. Selecting the stirrup **Reinforcement Layer** will open a **Property Window** showing the layers properties.
    c. In this example we want to modify the stirrup spacing, to do this click the >>> button behind **Edit Stirrup Distance.**



    d. The window for **Stirrup zones** appears.
    e. Click on the **[New Part]** button to enter a new section of reinforcement.
    f. Change the input type to **distance + total distance.**
    g. Change the **Distance [in]** field for this new section from **12"** to **4"** and chance **Total Distance [in]** to **36".**
    h. Under the heading **Minimum stirrup reinforcement,** change the **Distance to begin [in]** to **2".**

Note that the stirrups are placed closer together at both ends because the sections are automatically defined as Symmetrical.



i. After you are done, click **[OK]** to confirm your input.

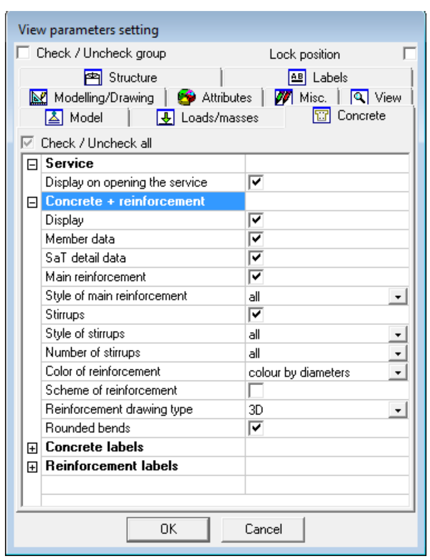
12. The graphical representation of the practical reinforcement can also be modified so that a 3D view of the reinforcement can be obtained. To do this, complete the following steps:

a. Right click anywhere in the workspace.

b. In the menu that has opened, select the Set view parameters for all option. The window for **View parameter settings** will appear.

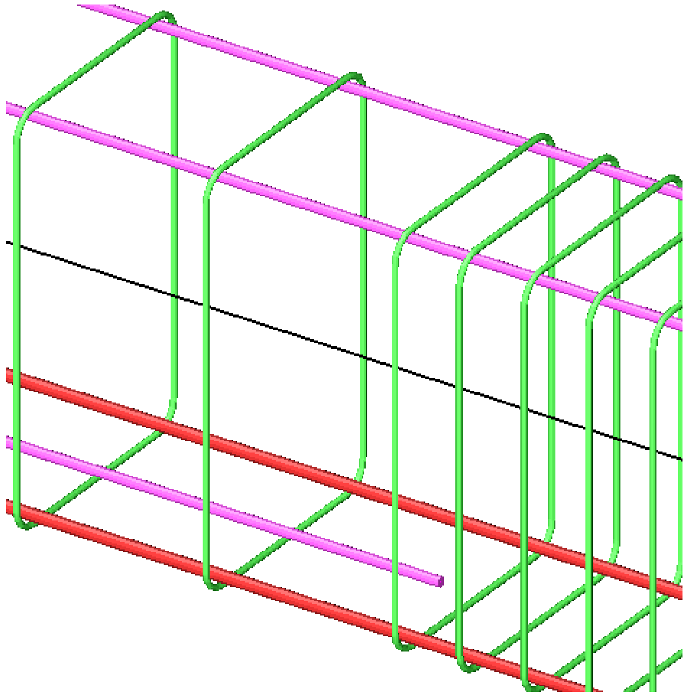
c. On the Concrete tab, the data and view options are displayed under the Concrete + reinforcement heading. Modify the view parameters in the following way:

- Set the option **Style of stirrups** to **All.**
- Modify **color of reinforcement** to read **color per diameters**.
- Set the **Reinforcement drawing type** to **3D**.
- Tick on the **Rounded bends** option.



d. Confirm your input by clicking **[OK].**

After modifying the necessary view settings, a 3D representation of the practical reinforcement is shown.



13. It is also possible to show a bill of reinforcement, which would contain a list of all the practical reinforcement on the project as well as information like the total length, weight and number of bars. To view the Bill of reinforcement, selection the icon Bill of reinforcement which can be found in the concrete menu. Once double clicking on the icon, select the **Preview** button in the properties window to display the following table.

Bill of reinforcement

Selection : All
The length of reinforcement and stirrups are calculated without rounded bends.
Type of position number : Global

Member	Position number	Size number	Diameter [inch]	Material	Length [inch]	Number of bars	W60 length [inch]	W60 weight [kips]
B3	1	3	0.375	W60	114.377	33	3774.440	0.118
B3	2	5	0.625	W60	240.000	2	480.000	0.042
B3	3	6	0.750	W60	240.000	2	480.000	0.060
B3	4	5	0.625	W60	160.000	1	160.000	0.014
				No.3(0.375inch)			3774.440	0.118
				No.5(0.625inch)			480.000	0.042
				No.6(0.750inch)			480.000	0.060
				Total for material			4894.440	0.234
				Total			4894.440	0.234

Note:

Automatic member design, including reinforcement design could serve as an alternative to manual input of reinforcement

Engineering Report

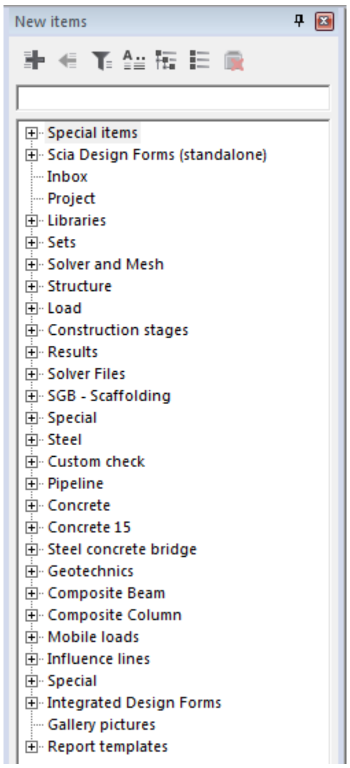
In this final part of the tutorial, the steps for how a calculation package can be set up and the different data sets that can be included in the Engineering Report will be discussed.

Formatting the Report

1. Double-click Engineering report in the **Main Window** or click in the button bar. The **Engineering Report** appears.

   The Engineering Report will open in a separate window.

2. Click the **[Insert]** button to show the New Items list. Each category can be expanded to show additional data types that can be added to the report. Each item can be added by double clicking on the item or using the **[+]** button.



3. By means of this window, data can be added to the report.

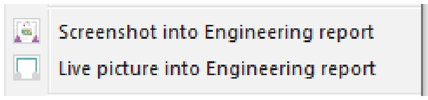
   - Open the **Libraries** group and double click on **Materials**.
   - Double Click **Cross-Sections**.
   - Open the **Structure** group and double click **Members**.
   - Open the **Results** group and double click **Internal forces on beam.**
   - Also, double click **Reactions** to add this item to the report.

The items that were added to the report are displayed in the **Navigator**. Drag or use the move up/down buttons to change the order of the items.

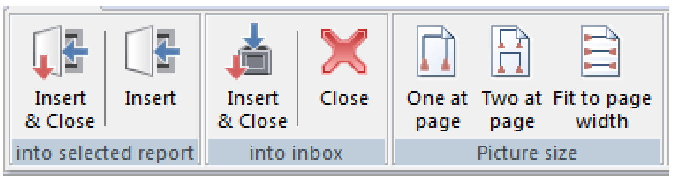
<u>Note:</u> For results, the properties table in the Engineering Report functions in the exact same way as it does when viewing results graphically. To see a specific result table, set the properties of the result and click the one of the regenerate buttons.

Adding an image to the document

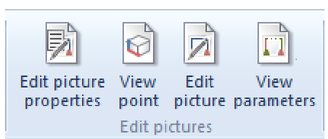
1. In the graphical interface (window) set the view of the model which you want as a picture in the Engineering Report.
2. Right click in the workspace and choose Screen shot into Engineering Report or Live Picture into Engineering Report.



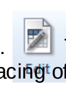
3. The insert window will open with properties that can be set specifically for the image. Once the settings are set as desired, select one of the import options to insert the picture either directly into the report or into the report inbox.

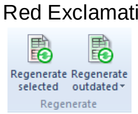


Note: The main differences between a Screenshot and Live Picture are that the Live Picture is a dynamic view of the model, meaning that if the inputs or results of the model change, the Live Picture will also update automatically. Additionally, since the image is dynamic, users have the opportunity to manipulate the results of the Live Picture using the Edit Picture buttons on the ribbon.



Additional Engineering Report Functionality

- Modification of Tables
  - Select specific table in Navigator and click "Edit" button.  This enables the user to select and modify specific items in the table, change the table layout and also the size and spacing of the table items.
- Refresh of Engineering Report
  - After adaptations to data or (content of) tables: Red Exclamation Point means item regeneration needed.
  - Regenerate Selected Items
  - Regenerate Outdated Items (Pictures, Tables)



- Properties of the different components
  - After selection of a component in the Engineering report Navigator, its properties can be consulted and adapted in the Properties menu.
- Combination key: display of governing linear combinations
  - New items > Sets > Combination key
  - Example: Take a look at Internal forces on beam, according to Combinations = ULS; Deformations on beam, according to Combinations = SLS. In these tables with results is referred to ULS/1 etc., and SLS/2 etc. The numbers after the combination names refer to the Combination key, where the governing linear combinations are written out.
- Engineering Manager gives you the option to have multiple Engineering Reports in the same Scia file. You can also have the Engineering Report open in the separate window while working in the Scia file. Any changes made to the Scia file can then be updated by regenerating the outdated items (marked by a red exclamation point) in the Engineering Report.
- The Engineering Report also allows for any information in the "Navigator" to be saved as a report template. These templates are found in the "Report Templates" item in the List of Item.